



Introduction to PanelView Programming using FactoryTalk View Studio (Machine Edition) Software





Notice:

The information contained in this manual contains excerpts from Rockwell Automation user manuals and is not meant to replace the complete documents. As such, this manual may not contain all the safety precautions or all of the information necessary to configure your automation system that is available in the complete user manuals. Please refer to the full product installation and user documentation, paying particular attention to all safety precautions prior to implementation.

The examples shown in this manual are for educational purposes only. Any individual or company utilizing these examples in a control system does so at their own risk and agrees to hold Van Meter Industrial harmless from any loss, accident, or injury resulting from the use of these examples.

All Rockwell Automation information included in this manual is reproduced with the permission of Rockwell Automation, Copyright © 2002 - 2012 Rockwell Automation. All rights reserved.

Chapter 1: Introduction	4
Course Objective.....	4
Development Steps	4
Important Concepts.....	4
Navigation Reference / Stops	5
Hardware and software for this course	6
Chapter 2: Lab Exercises	7
<i>Lab Exercise 1: Changing a PC's IP address</i>	<i>7</i>
<i>Lab Exercise 2: Changing the IP address of a PanelView Plus.....</i>	<i>11</i>
<i>Lab Exercise 3: Configuring RSLinx Classic</i>	<i>16</i>
<i>Lab Exercise 4: Creating a PLC program (Live demo + Class participation).....</i>	<i>20</i>
<i>Lab Exercise 5: Creating an Application</i>	<i>21</i>
<i>Lab Exercise 6: RSLinx Enterprise Communications.....</i>	<i>24</i>
<i>Lab Exercise 7: Creating an Object</i>	<i>29</i>
<i>Lab Exercise 8: Creating an Indicator</i>	<i>39</i>
<i>Lab Exercise 9: Numeric Data Display</i>	<i>47</i>
<i>Lab Exercise 10: Numeric Input Enable.....</i>	<i>54</i>
<i>Lab Exercise 11: Saving/Downloading a Program.....</i>	<i>63</i>
<i>Lab Exercise 12: Creating a Bar Graph.....</i>	<i>71</i>
<i>Lab Exercise 13: Creating a Gauge</i>	<i>83</i>
<i>Lab Exercise 14: Working with Basic Shapes</i>	<i>90</i>
<i>Lab Exercise 15: Working with Graphics.....</i>	<i>95</i>
Using Libraries	95
Using Symbol Factory.....	96
Importing Graphics.....	102
<i>Lab Exercise 16: Changing Screens</i>	<i>107</i>
Advanced Labs	118
<i>Advanced Lab Exercise 1: Alarms.....</i>	<i>118</i>
<i>Advanced Lab Exercise 2: Application Backup and Restoration</i>	<i>125</i>
<i>Advanced Lab Exercise 3: Uploading a Program.....</i>	<i>133</i>
<i>Advanced Lab Exercise 4: Restoring Runtime Applications</i>	<i>136</i>
<i>Advanced Lab Exercise 5: Resizing the Display.....</i>	<i>140</i>
<i>Advanced Lab Exercise 6a: Factory Talk Users and Groups</i>	<i>145</i>
<i>Advanced Lab Exercise 6b: Screen Security.....</i>	<i>153</i>
<i>Advanced Lab Exercise 7: Global Objects</i>	<i>164</i>
<i>Appendix A: Helpful Links</i>	<i>172</i>

Chapter 1: Introduction

Course Objective:

Provide the user with the basic skills necessary to operate, maintain, and develop applications for the PanelView Plus human machine interface. These include, but are not limited to: creation and implementation of various interface objects, application development, and handling of data received from an external source. At the conclusion of this course, the user should be familiar with all these tasks and have basic knowledge to implement these skills in their respective applications.

Development Steps

This course contains many tasks that are to be completed in a specific order. When this is necessary, a text box will be present with the step number indicated on the left-hand side. These steps should be completed in logical numeric order, counting up from 1 (i.e. 1, 2, 3, ...). An example of this box, indicating the first step in a series, can be seen below.



Important Concepts

Important information that is relevant to the topic at hand will be denoted by an image of a light bulb within a colored text box with text explaining important concepts or other information. An example of such a text box can be seen below.

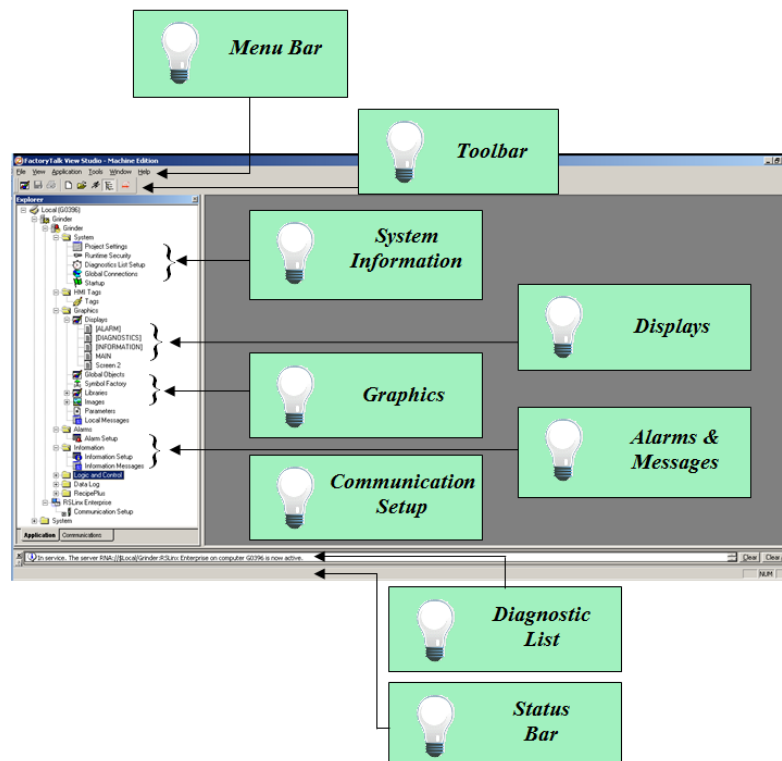


Stops

At the end of each lab, there will be a symbol signifying that you need to stop and wait for the instructor to tell you to proceed. During this time feel free to explore the software and practice what the previous lab had you do, also answer any review questions.



Navigation Reference



Hardware and software for this Course

Hardware:

- CompactLogix L3Y Demo case or CompactLogix L43 Demo case
- PanelView Plus 6 (Terminal size may vary)

Software:

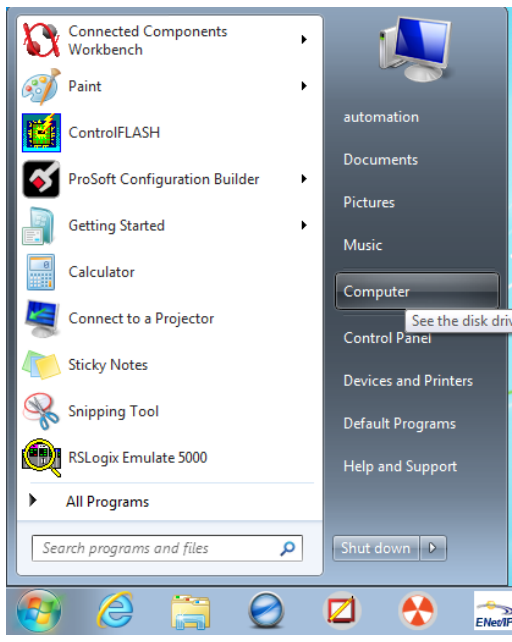
- Studio 5000 - Version 21.XX or RSLogix 5000 Version 20.XX
- FactoryTalk View Studio Machine Edition - Version 7.0 or 8.0
- Computer Operating System – Windows 7 64-bit Virtual Machine

Chapter 2: Lab Exercises

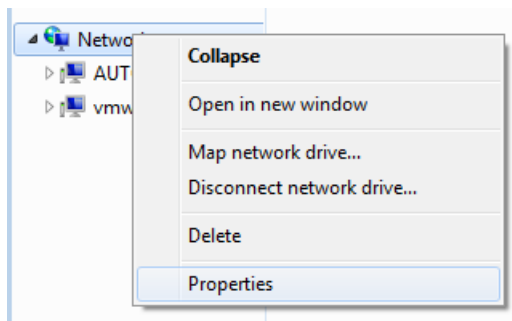
Lab 1: Changing your PC's IP address

Objective: Teach the user how to change the IP address of a given PC. This is a must when working with an Ethernet network. All devices that you want to network must be on the same Ethernet sub-network, but each device must have a unique address as there can be no duplicates.

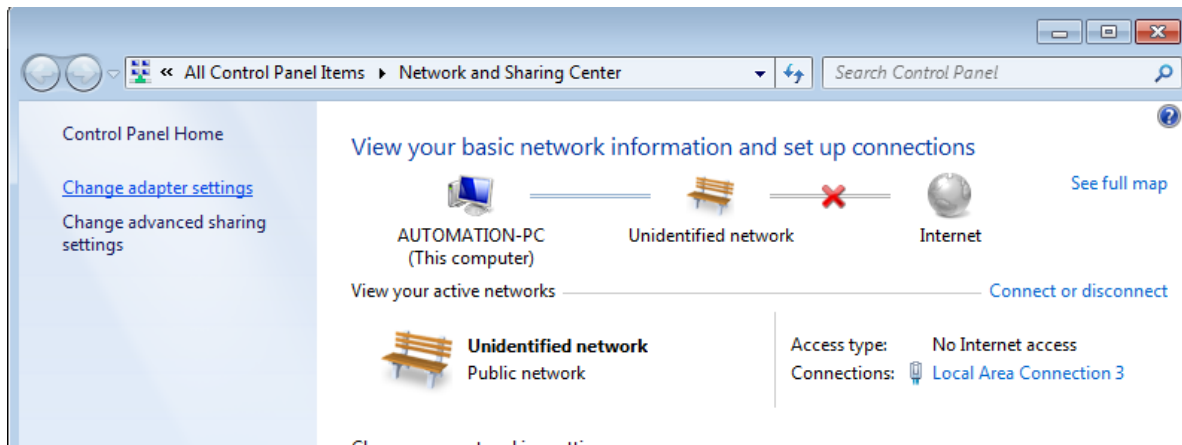
1. Click the start button from your desktop and select “My Computer” or “Computer”.



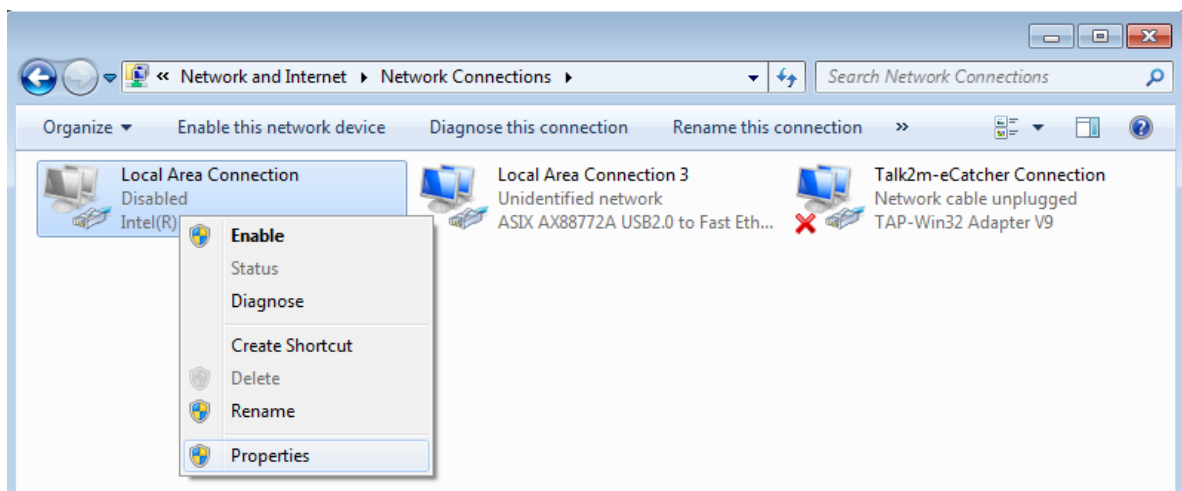
2. The computer window will open up. On the left side of the window, right-click on “Network” and select “Properties” from the bottom of the subsequent menu.



- After completing step 2, the “Network and Sharing Center” window will open. On the left side of that window, click “Change Adapter Settings”.

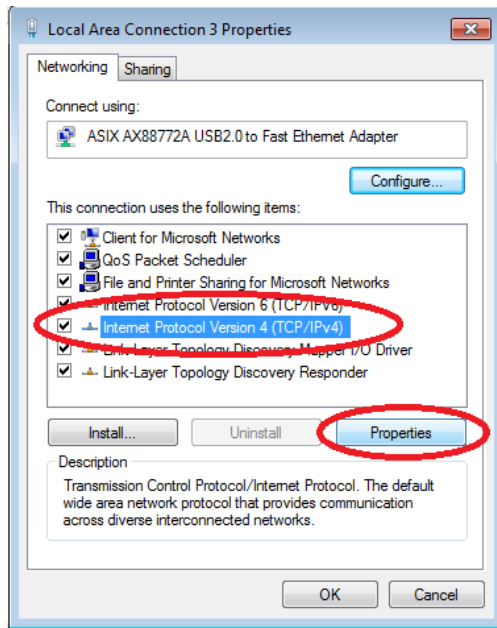


- After clicking on “Change Adapter Settings” from step 3, the “Network Connections” window will open. Right-click on “local Area Connection” and select “properties” from the bottom of the subsequent menu.

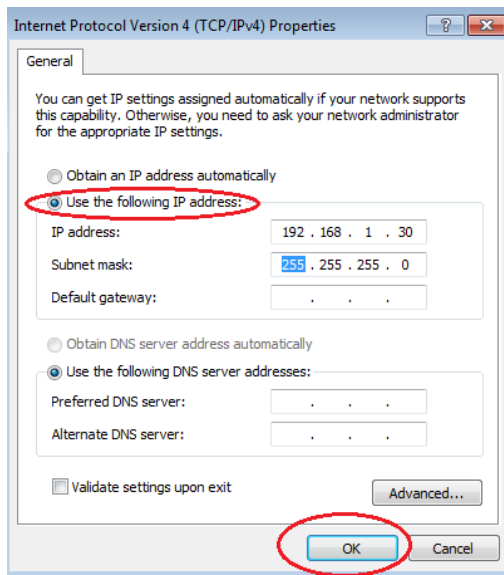


You may see other Ethernet connections in this window. The connections you are seeing are physical or wireless Ethernet connections available to your PC.

- Once you click the “properties” option from step 4, the “Local area connection properties” window will open. From the new window, highlight “Internet Protocol version 4 TCP/IPv4” and then click the “properties” button as shown below.



- After selecting “Properties” from step 5, the “TCP/IPv4” properties window will appear. From this window select “use the following IP address”, enter the IP address and subnet mask as shown below and select “OK” at the bottom of the window.



- Close all open windows until you are back at the Windows desktop.

Lab 1 Review:

1. What does a subnet mask do?
2. Can you have two devices with the same IP address on a given subnet mask?
3. What is a quick way to check the IP address of your PC (Hint: CMD)

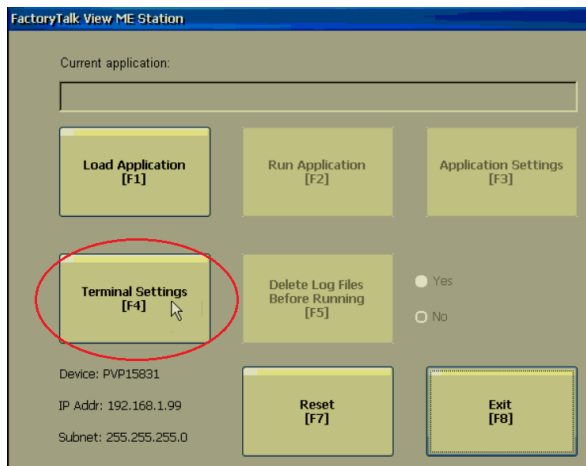


This completes Lab 1. Please wait for instructions to proceed further.

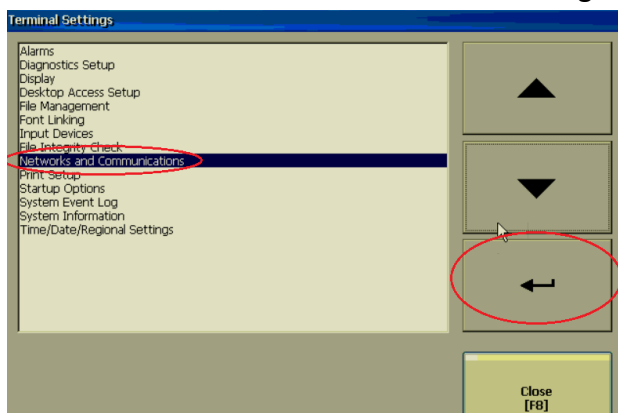
Lab 2: Changing the IP address of a PanelView Plus

Objective: To teach user how to change the IP address of a PanelView Plus terminal.

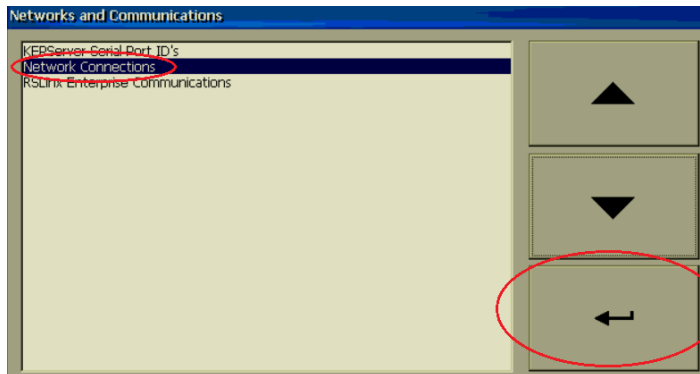
1. From the PanelView, FactoryTalk View ME Station should be open. Select the “Terminal Settings” button.



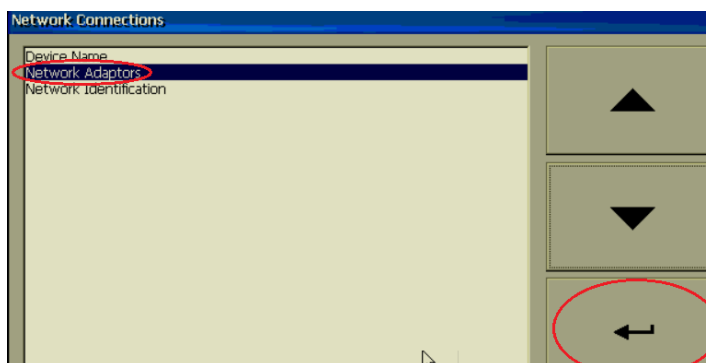
2. After selecting terminal settings from step 1, scroll down to “Networks and Communications” within the terminal settings window and press the “Enter” button.



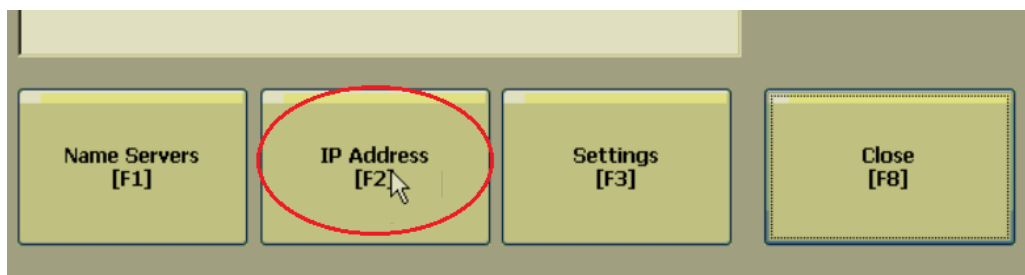
- After selecting “Networks and Communications” from step 2, the Networks and communications window will open. Scroll down to “Network Connections” within the list and press the “Enter” button.



- After selecting “Network Connections” from step 3, the Network Connections window will open. Scroll down to “Network Adaptors” within the list and press the “Enter” button.



- After selecting “Network Adaptors” from step 4, the network adapters window will open. Press the “IP Address” button at the bottom of the window.



6. After selecting “IP Address” from step 5, the IP address window will open. Press the “IP Address [F1]” button.

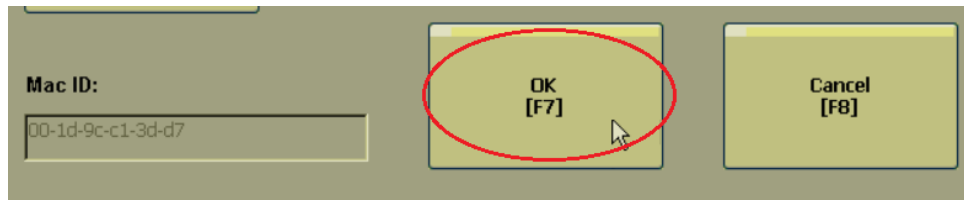
The screenshot shows a window titled "IP Address" with a blue header bar. Inside, there are three input fields on the left: "IP Address [F1]" (circled in red), "Subnet Mask [F2]", and "Gateway [F3]". The "IP Address" field contains "192.168.1.99", "Subnet Mask" contains "255.255.255.0", and "Gateway" contains "0.0.0.0". To the right of these fields is a "Use DHCP [F4]" button with two radio buttons: "Yes" (unselected) and "No" (selected). At the bottom left is a "Mac ID:" field with the value "00-1d-9c-c1-3d-d7". At the bottom right are "OK [F7]" and "Cancel [F8]" buttons.

7. After pressing the “IP Address” button from step 6, the IP address window will open. Enter in an IP address of **192.168.1.10**, then press the “Enter” button at the bottom of the window

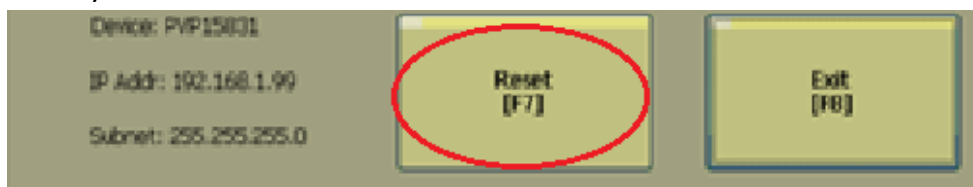
This screenshot shows the same "IP Address" window as in step 6, but with a virtual keyboard overlay. The "IP Address" field now contains "192.168.1.99" (circled in red). The virtual keyboard has a numeric keypad at the top, followed by rows of letters (q-w-e-r-t-y-u-i-o-p, a-s-d-f-g-h-j-k-l-semicolon, z-x-c-v-b-n-m-comma), and a bottom row with "Select F1", left and right arrow keys, up and down arrow keys, an "Enter" button (circled in red), and an "ESC" button. The "Use DHCP" section and "OK/Cancel" buttons are also visible.

8. Repeat step 7 for the subnet mask, except enter a value of “255.255.255.0”. If this value is already present, skip this step.

9. After setting the IP address in steps 7 & 8, press the okay button at the bottom of the window. Click “Okay” on any subsequent warnings, then press the “Close” button at the bottom of each window until you are back at the main “FactoryTalk View ME Station” window from step 1.



10. From the main FactoryTalk View ME Station window, select the “Reset” option at the bottom of the window. You will be prompted to reset the terminal at this point, click yes or okay to reset the PanelView.



At this point the panelview will shut down, and reboot. This is normal operation, as we made changes that will need a power-cycle to take effect.

Lab 2 review:

1. How could you check the firmware revision running on the PanelView?
2. Can I change what application loads upon start-up, from the PanelView?
3. Fill in the blanks for the path to set the PanelView's IP address: Terminal settings→Networks and Communications→()→()→()

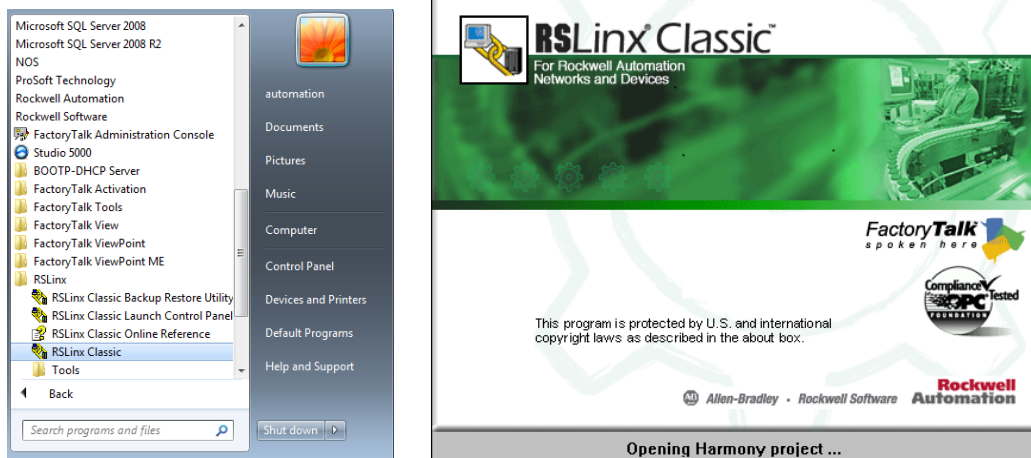


This completes Lab 2. Please wait for Instructions to proceed further.

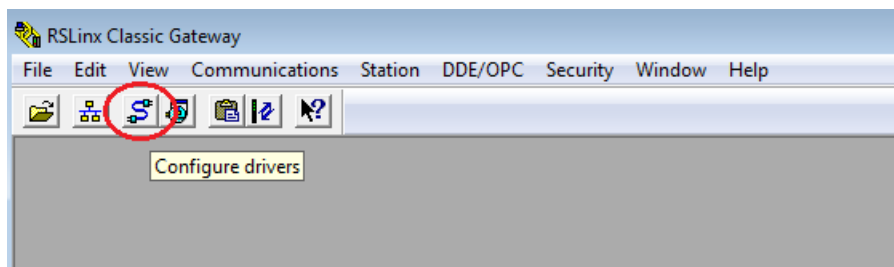
Lab 3: RSLinx Classic Ethernet/IP Driver Setup

Objective: To teach the user the differences between Ethernet drivers, how to navigate RSLinx classic and how to utilize RSLinx Classic with Studio 5000 / FactoryTalk View Studio.

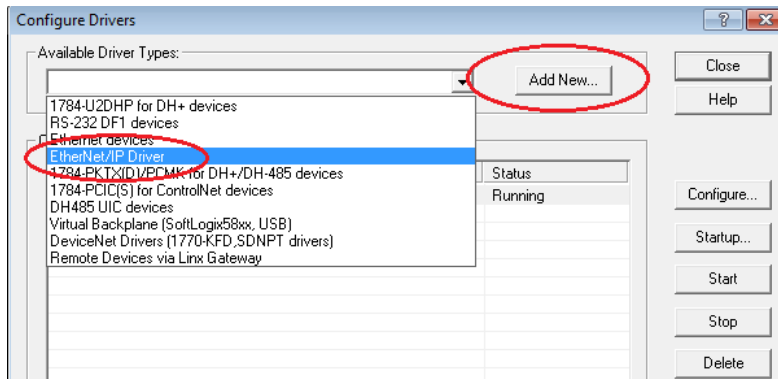
1. From the start menu navigate to the following location: “Start→All Programs→Rockwell Software→ RSLinx→RSLinx Classic”. Select “RSLinx classic” and it will open after a few moments.



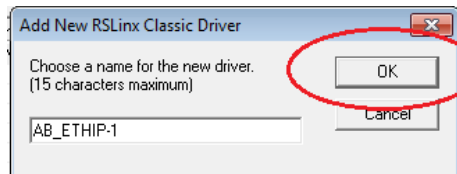
2. Once RSLinx classic opens, Select the “Configure Drivers” button from the toolbar, as we will need to create an Ethernet driver.



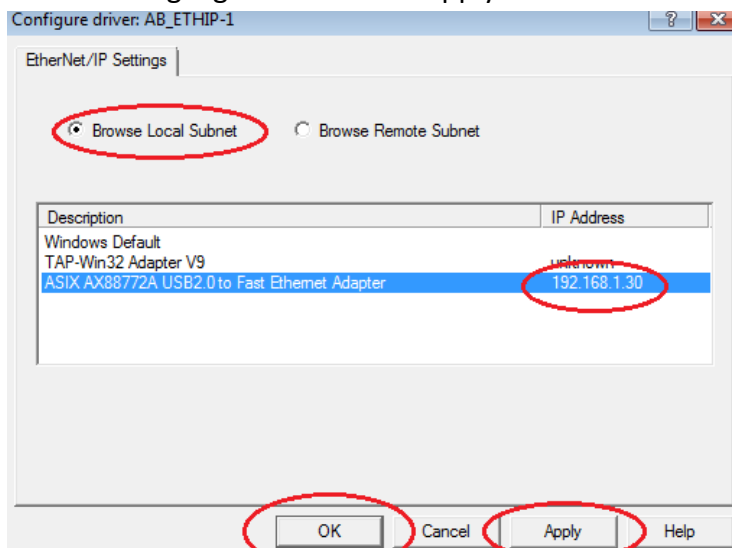
- Once you have selected configure drivers from step 2, select “Ethernet/IP driver” from the drop down menu of the configure drivers window. Once you have that driver highlighted, click the “Add New” button.



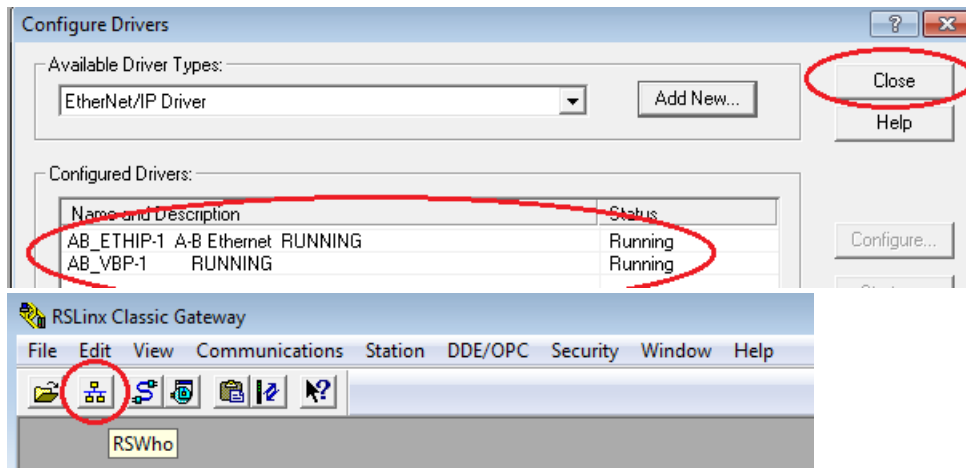
- Leave the default name of “AB_ETHIP-1” when you are asked to name the new driver. Click the “Okay” button.



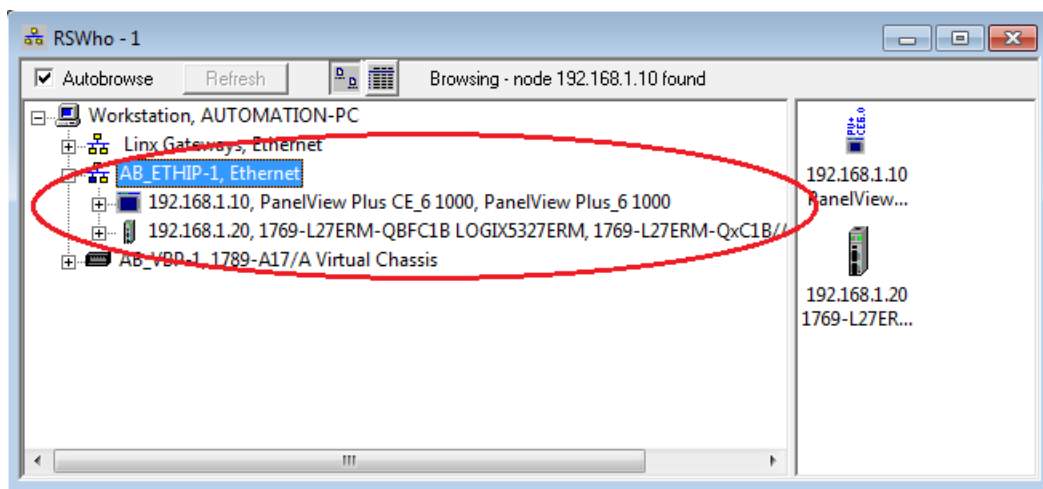
- After clicking okay from step 4, the Ethernet/IP Settings window will open. From here we need to make sure “Browse local subnet is selected”. We also need to make sure that the IP address of our PC is highlighted as the Ethernet adapter to use. Once the IP address is highlighted click the “Apply” then “OK” buttons at the bottom.



6. You will now notice that your newly created Ethernet/IP driver is running. Click the “Close” button and then click the “RSWho” button to start browsing for devices.



7. Once the RSWho window is open, click on the “+” next to the AB_ETHIP-1, Ethernet” driver. You should now be browsing the subnet of 192.168.1.XXX and you should also see your PanelView with the IP address 192.168.1.10 and your controller at 192.168.1.20.



Lab 3 Review:

1. What is the difference between the Ethernet Devices and Ethernet/IP driver?
2. What driver in RSLinx would you use if you wanted to communicate to a processor via serial?
3. Fill in the following IP Address information:
 - Computer IP Address :
 - Controller IP Address :
 - PanelView IP Address :



This completes lab 3. Please wait for instructions to proceed further.

Lab 4: Creating a PLC program & I/O configuration

1. This lab will consist of a live demo by the instructors. We will look briefly at the demo code created for this lab.

Lab 4 review

1. How would someone look up the firmware revision of an I/O module?
2. Where do I establish the communication path between my PLC and PanelView?

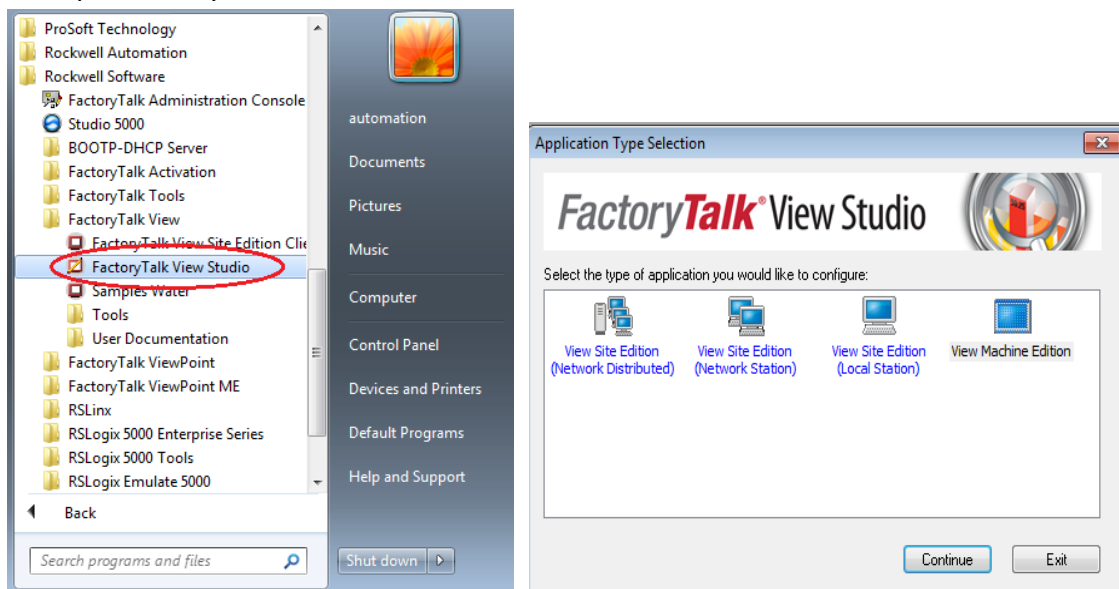


This completes lab 4. Please wait for instructions to proceed further.

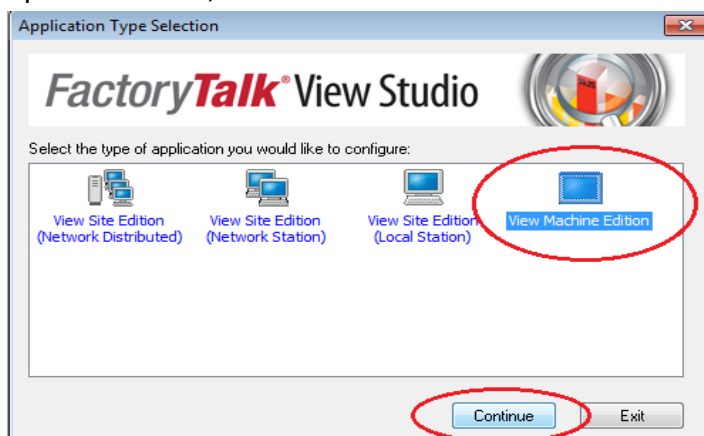
Lab 5: Creating an application

Objective: To familiarize the user with basic FactoryTalk View Studio operation, such as opening the program and creating a new application. This will enable the user to create the specific PanelView application they will configure in the following labs.

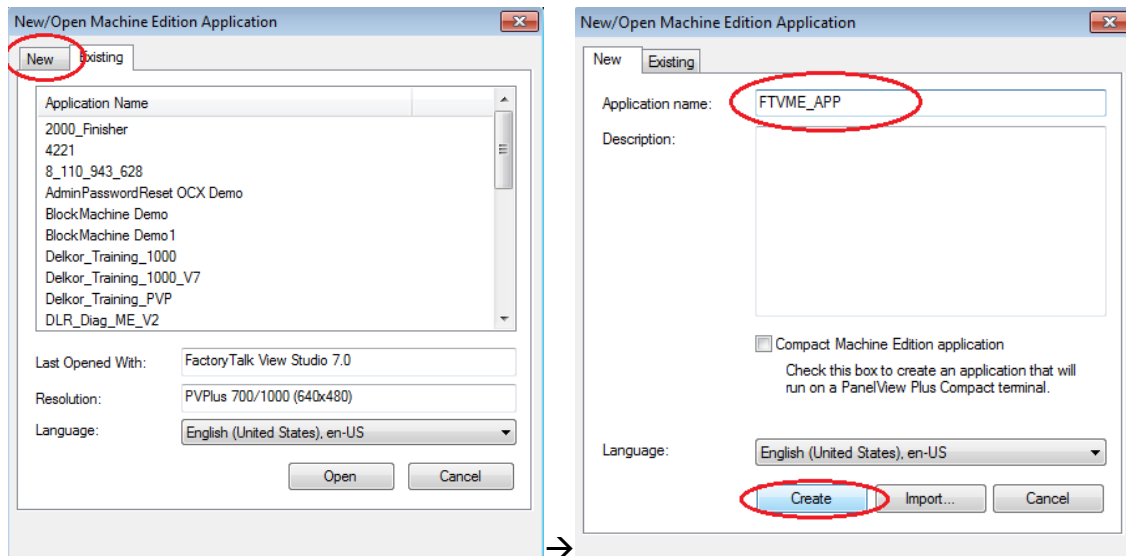
1. We will start by opening FactoryTalk View Studio from the following location. “Start→All Programs→Rockwell Software→Factory Talk View→ Factory Talk View Studio”. Once you have navigated to that location, click “Factory Talk View Studio and the software will open shortly after.



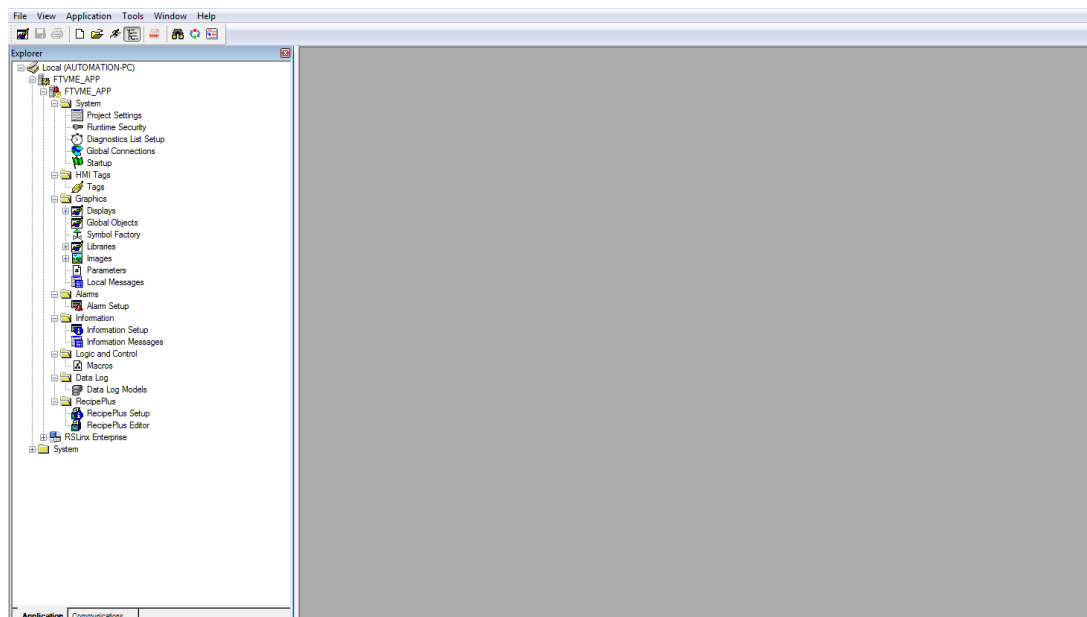
2. Once the software has opened, select “View Machine Edition”. Once you have this option selected, click the “Continue” button at the bottom of the window.



- After selecting “Continue” from step 2, you will be brought to the “New/Open Machine Edition Application” window. Select the “New” tab so that we can create a new application. After you click the “New” tab, there will be a brief loading period and then you will be asked to name your application. Give you application the name of “FTVME_APP” then click “Create” at the bottom of the window.



- After you click “create” from step 3, there will be a brief loading period. This is the time period where the project is created in FactoryTalk View Studio. After the Application is created, you will notice that the project explorer on the left side of the window has now been pre-configured.



Lab 5 Review

1. Where would you find the communication settings between the PLC and PanelView?
2. Is there a way to configure security for screens and users?
3. How do you create a new display?

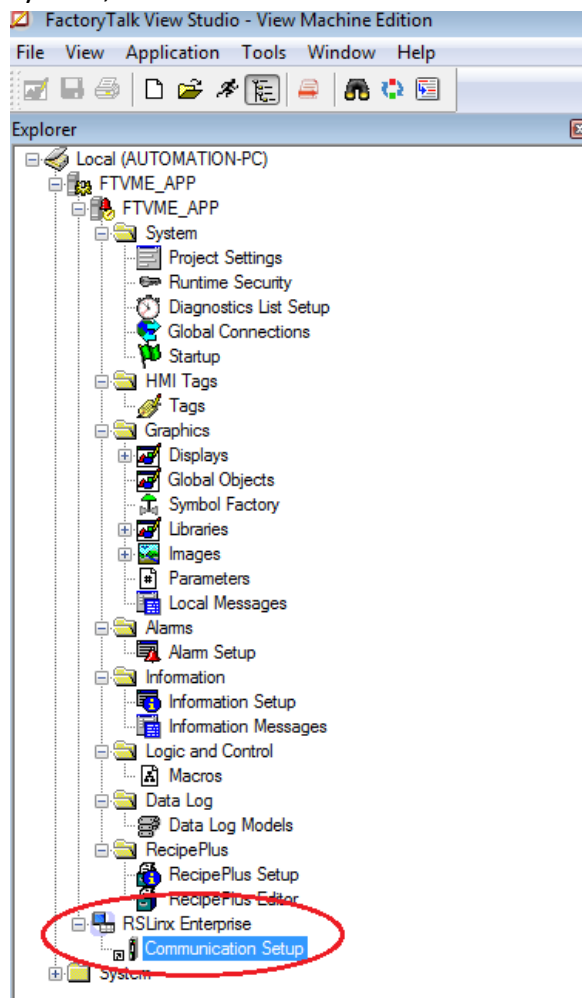


This now completes lab 5. Please wait for instructions to proceed further.

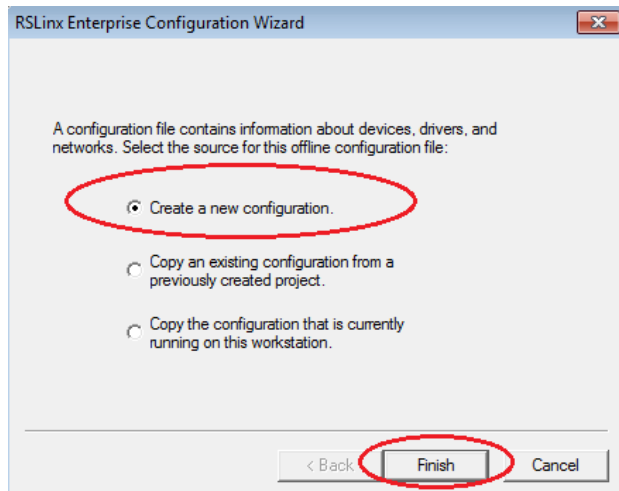
Lab 6: RSLinx Enterprise Communications Setup

Objective: Teach the user about configuration of RSLinx Enterprise Communications. This allows for downloading and uploading of user-created applications over a network. This also established the link or “path” between a controller and PanelView. At the end of this lab, the user will have configured their PanelView to communicate easily with the PLC.

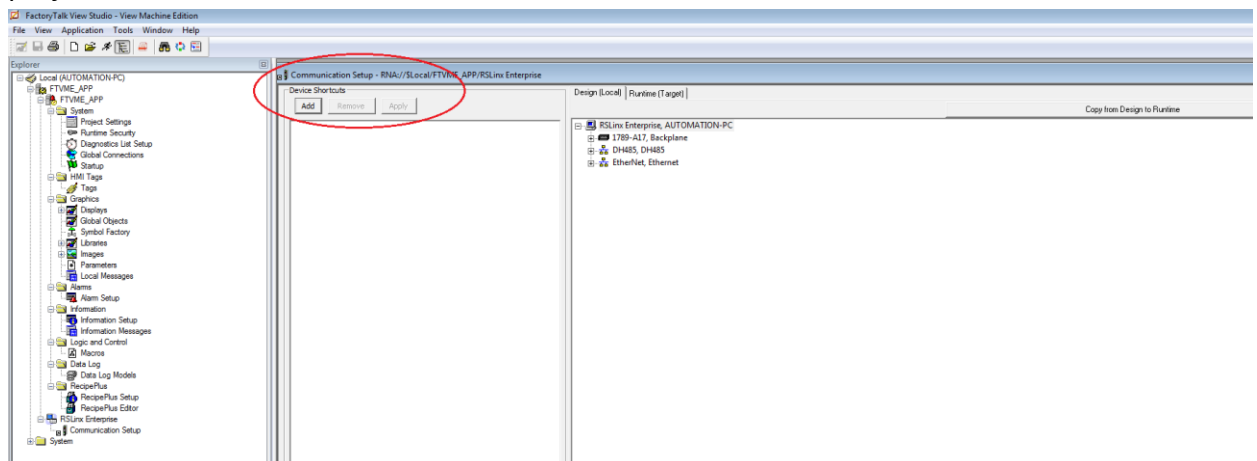
1. The first thing we need to do for configuration of RSLinx Enterprise is to click the “+” next to RSLinx Enterprise in the project explorer tree. Once you have clicked the “+” symbol, double-click on “Communication Setup”



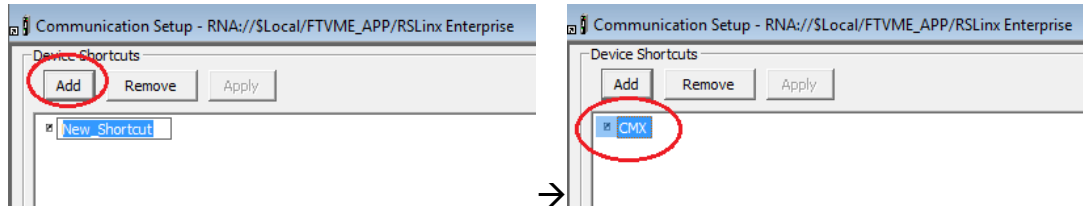
2. Since this is a new project, you will be prompted to run the RSLinx Enterprise Configuration Wizard after double clicking on “Communication Setup” in step 1. When the wizard opens, select “Create a new configuration” and then click “Finish”



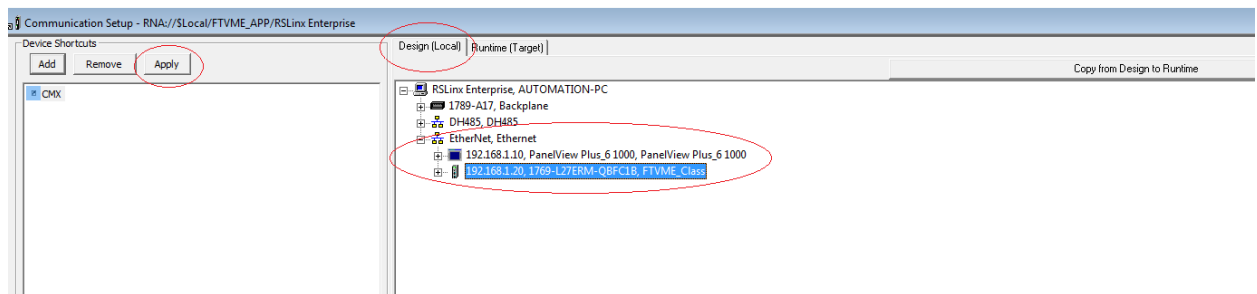
3. After clicking “finish” from step 2, the communication setup window will open. Once this opens, the first thing we need to do is add a device shortcut to our PLC under where is says “Device Shortcuts”. There are currently none in there because this is a new project.



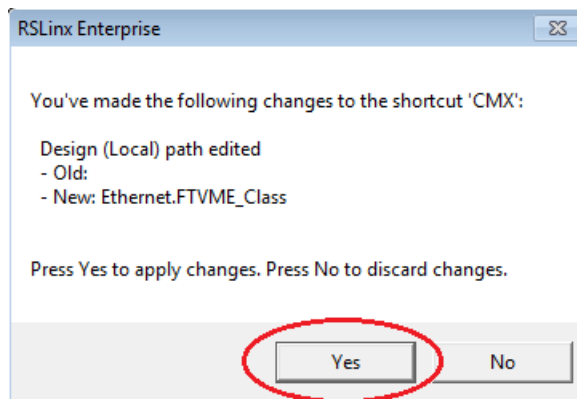
4. From the “Device Shortcuts” side of the window, click the “Add” button. Once you click add, you will be prompted to name your device shortcut. For this lab, name the shortcut “CMX” and then press the “Enter” key on your keyboard. The new shortcut should look like the image below.



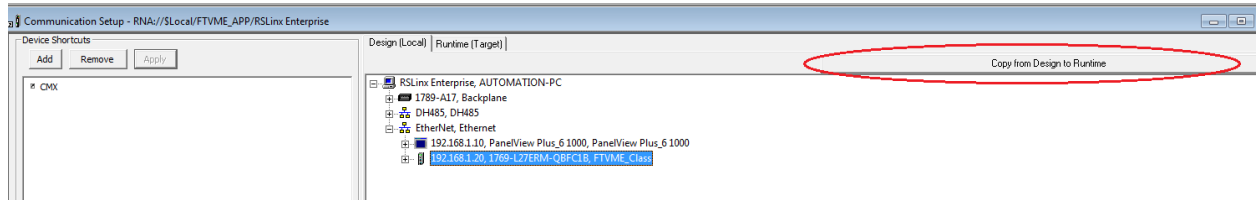
5. Now that we have created a device shortcut, we need to associate that with a PLC on the network. Jump over to the “Design [Local]” side of the window, click the “+” next to the “EtherNet,Ethernet” driver, highlight the IP address of your PLC, then jump back over to the “Device Shortcut” side of the window and click the “Apply” Button. This associated our shortcut name with our PLC.



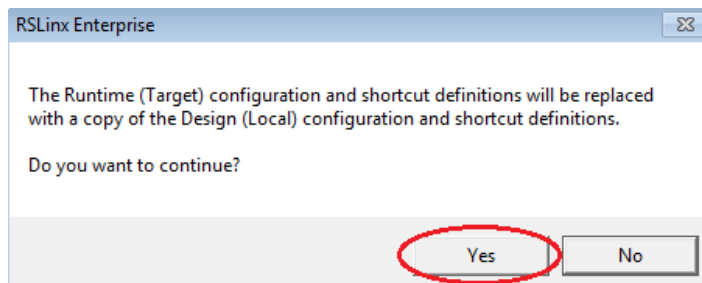
6. After clicking “Apply” from step 5, you will get an informational message stating the changes you have made. Click “Yes” within the subsequent message.



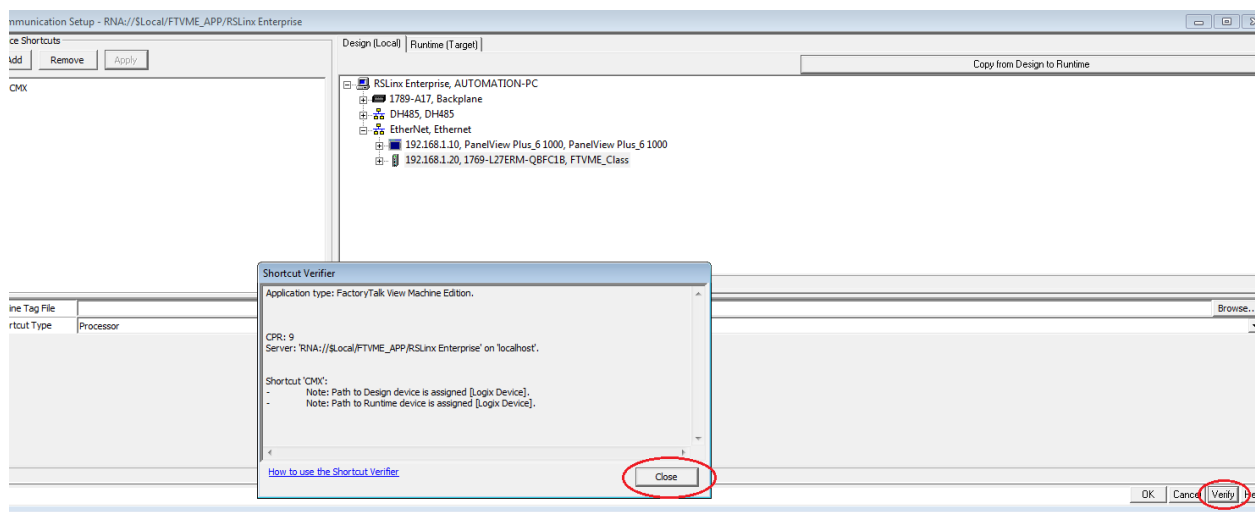
7. The next thing we need to do is associate the path we just created in the “Design[Local]” tab, with the configuration in the “Runtime[Target]” tab. This makes sure that when we download the application to the PanelView, the communication path to our PLC goes along with that .MER (Machine Edition Runtime) file. This is done by clicking the “Copy from Design to Runtime” button at the upper left corner of the “Design[Local]” tab.



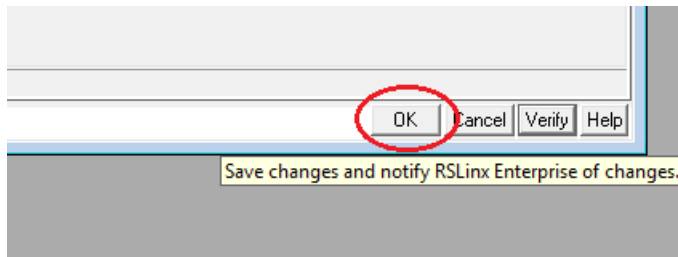
8. Once you click “Copy from Design to Runtime” from step 7, you will get a confirmation message stating the changes you have made. Click “Yes” on the subsequent message.



9. The next thing we need to do, just for good measure is to verify the changes we have made by clicking the “Verify” button show below. This shows us our current communication path. Click “Close” on the “shortcut Verifier” window.



10. This step is crucial!! Make sure you click the “OK” button near the verify button from step 9, otherwise the changes we made will not be saved! Once you click “OK” the RSLinx Enterprise configuration window will close, this is normal.



Lab 6 review:

1. What do you need to do to make sure that the communication path gets sent to the PanelView Plus at runtime?
2. Is it necessary to click the Verify and OK buttons at the bottom of the RSLinx Enterprise window, after assigning a shortcut?

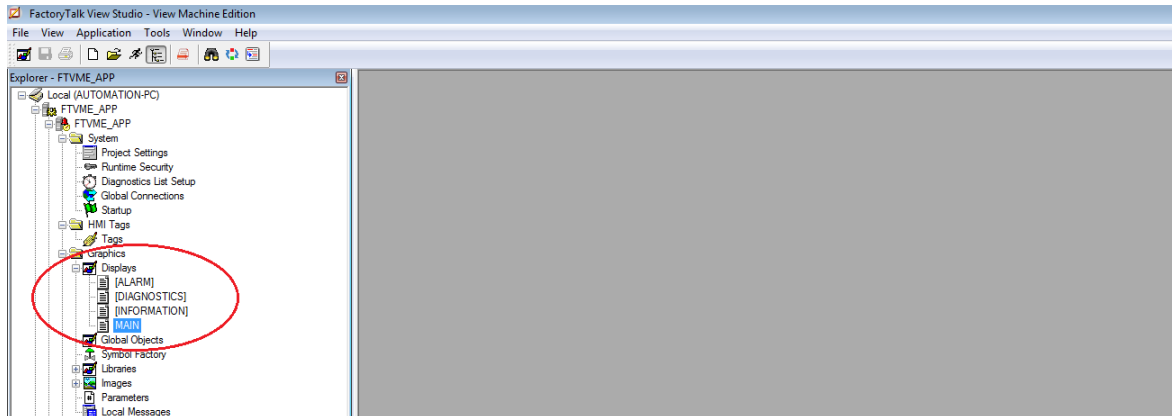


This completes lab 6. Please wait for instructions to proceed further.

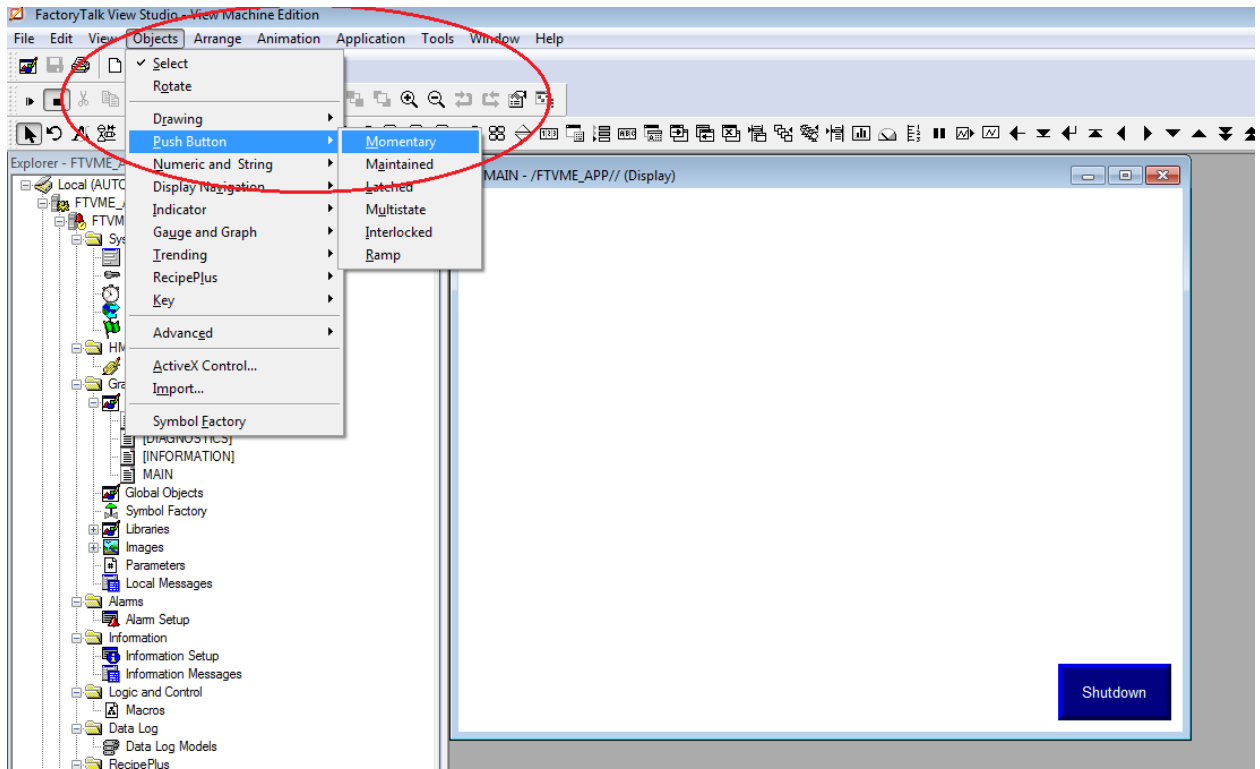
Lab 7: creating an object

Objective: To demonstrate the process of adding objects to the displays. In addition, the user will learn to manipulate objects, appearances, behaviors, and connections. By the end of the lab the user will have gained general knowledge in creating basic objects.

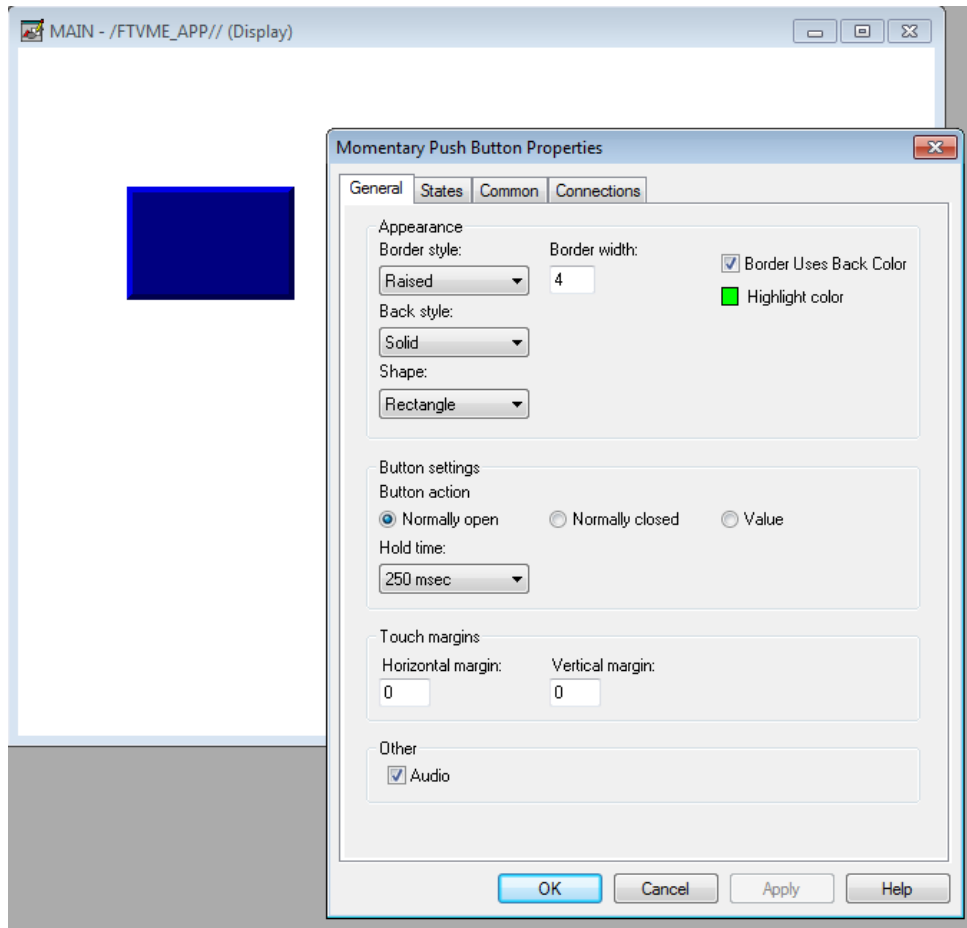
1. Start by clicking the “+” next to displays from the project explorer tree. Scroll down to the display titled “MAIN” and double click it. This will open the main display.



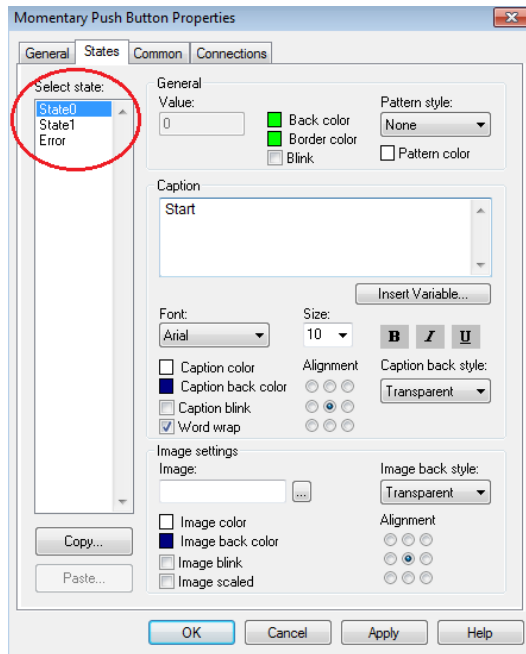
2. Once the main display opens, we will start by adding a momentary pushbutton to the screen. From the toolbar, select “Objects→Push button→Momentary”.



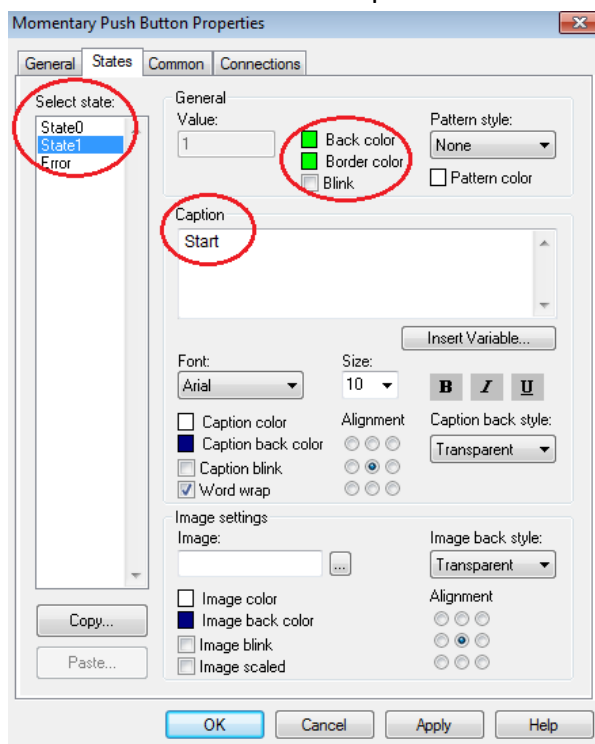
3. You will notice a change in the cursor, so now you can make a square on the screen by clicking and dragging the cursor until you have made your basic shape. Once you release the mouse, the momentary pushbutton properties window for that push button will open.



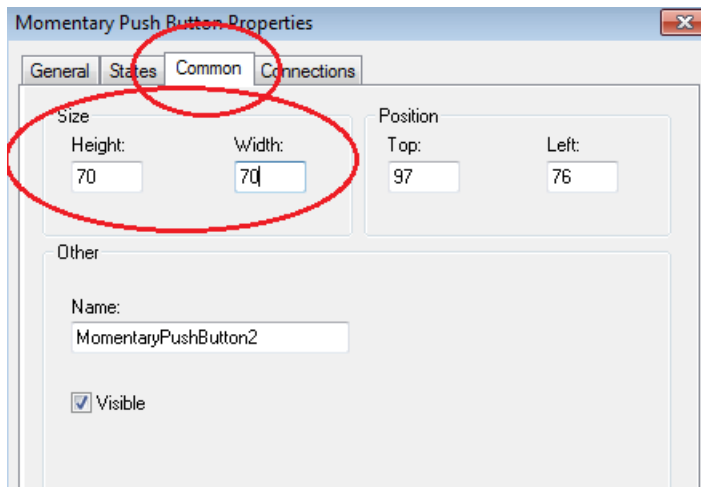
4. You will notice at the top of the properties window that there are 4 tabs: General, States, Common and Connections. Select “States” tab, leave the “select State” to state 0 and enter the following information in the image below.



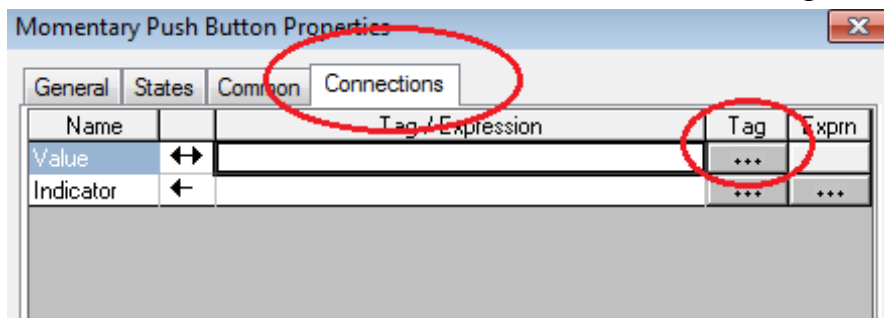
5. After making the changes from step 4, change the “state select” to state 1, and enter the same information as step 4.



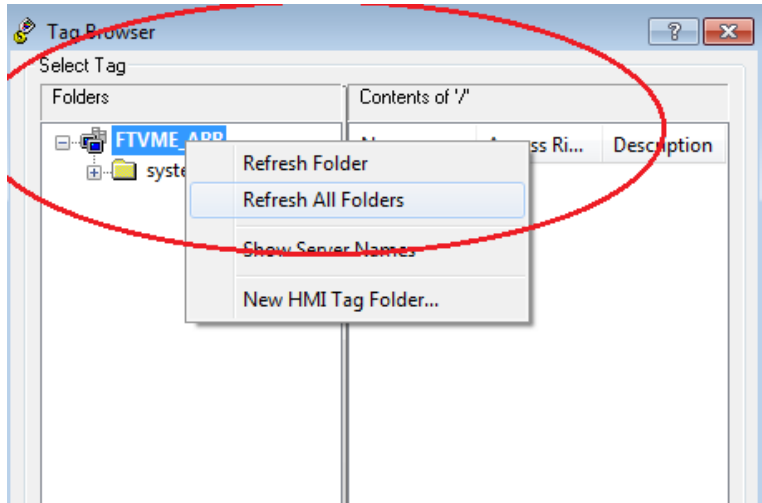
6. Once you have completed steps 4 & 5, change over to the “Common” Tab and enter a value of “70” for the height and width.



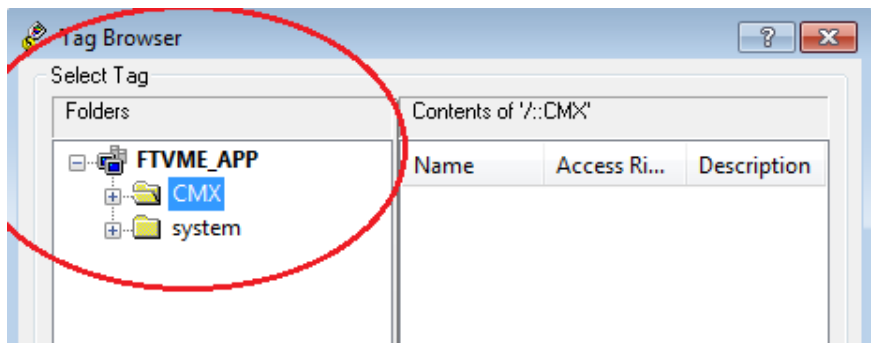
7. Up to this point we have created a push button, specified the size and color. The next step will be to link the push button to a tag in the PLC so that our PanelView can control actual I/O. Change over to the “Connections” tab. Once you have changed over to the connections tab, click the browse button as shown in the image below.



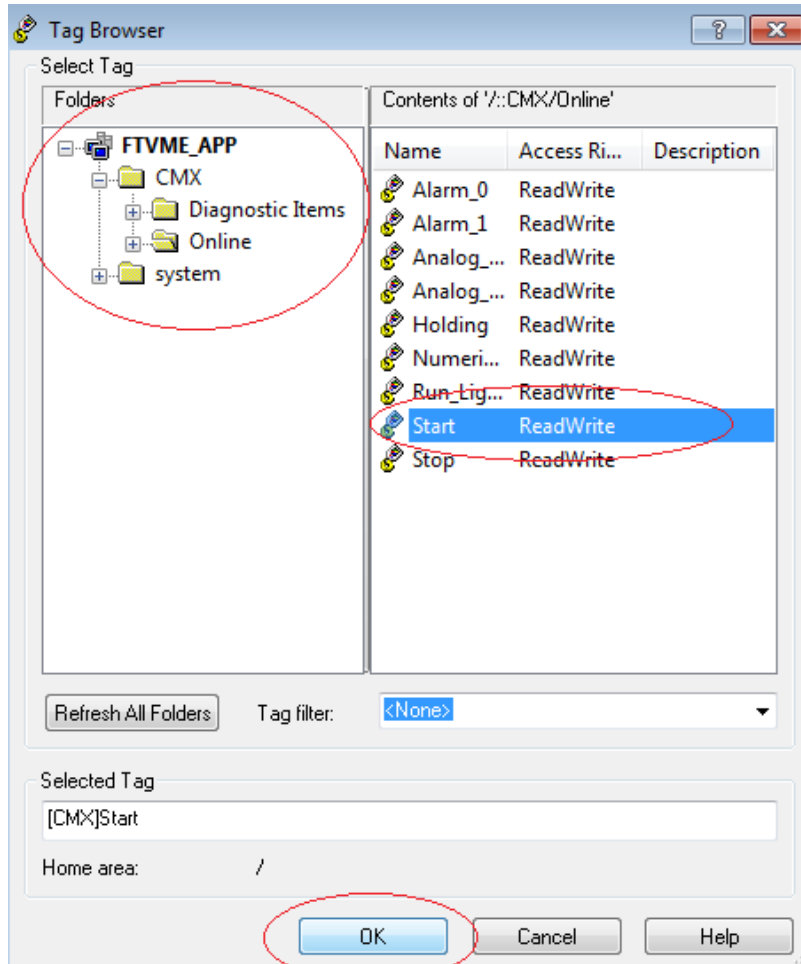
8. After clicking the browse button from step 7, a tag browser window will open. This is the window in which we will search for PLC tags to link to our objects. Start by right-clicking on the “FTVME_APP” icon underneath the folders portion of the tag browser window. From the subsequent menu, select “Refresh all folders” as shown in the image below.



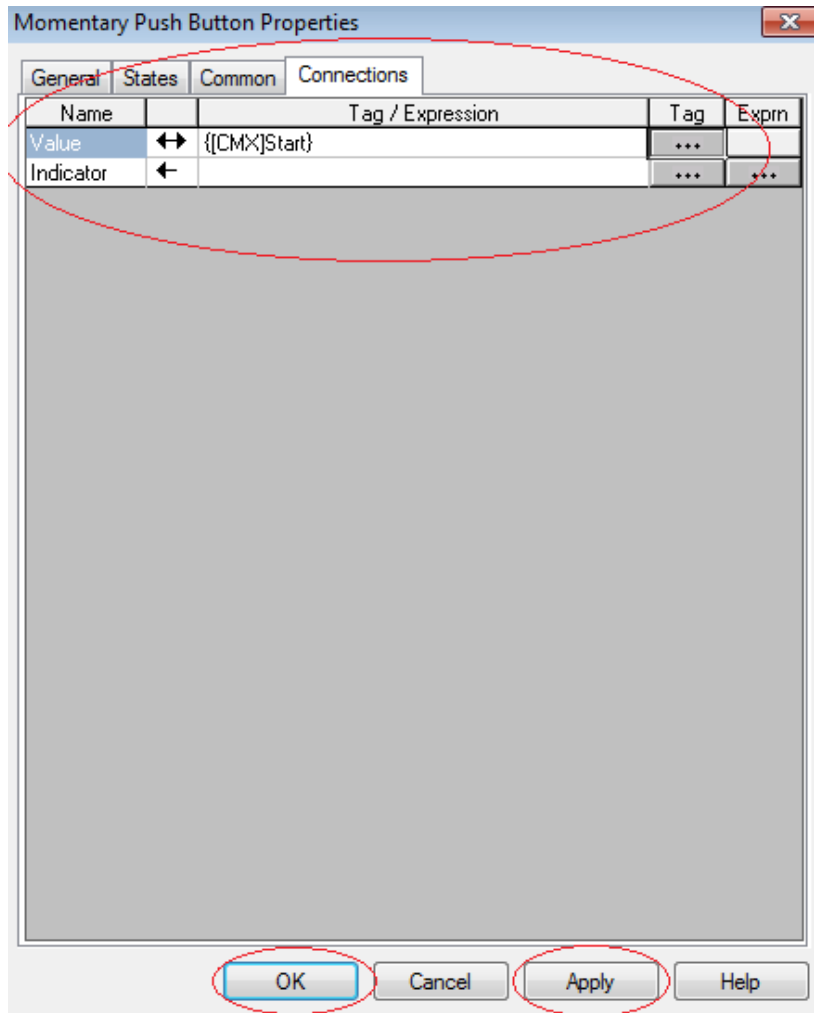
9. In step 8, the only folders underneath the “Folders” portion of the tag window were FTVME_APP and system. By clicking refresh all folders from step 8, we are updating Factory Talk view studio so that our controller project file, where our tags are stored, shows up. You should now see a “CMX” folder as well.



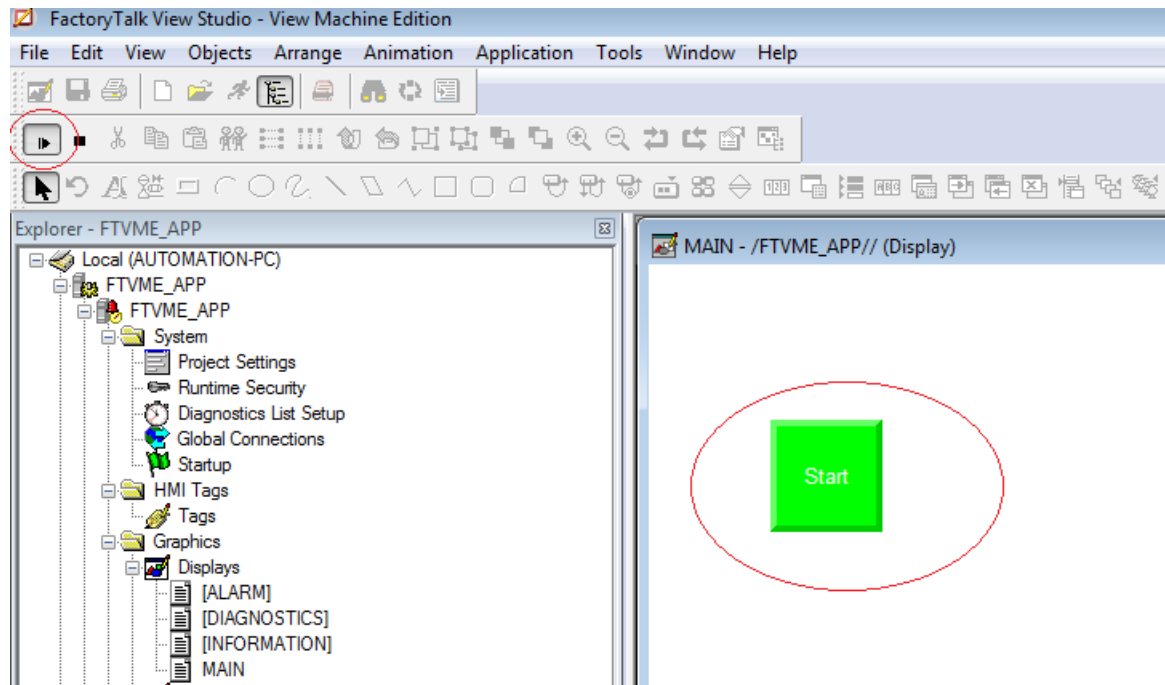
10. Now that our controller project file is available to browse for tags, click the “+” next to the “CMX” folder. The CMX folder will expand and once you highlight the “online” folder below it, you will be able to see our controller memory tags. Highlight the tag named “Start” on the list of tags and click “OK” at the bottom of the tag browser window.



11. By clicking “OK” at the end of step 10, you are assigning the tag in your controller named “Start”, to your newly created Start pushbutton. After clicking okay you will be brought back to the momentary push button properties for your start button, but this time you will notice the tag “{[CMX]Start}” in the value field. Click the “Apply” button at the bottom of the window first, then the “OK” button. This will close the properties window and bring you back to your main display.



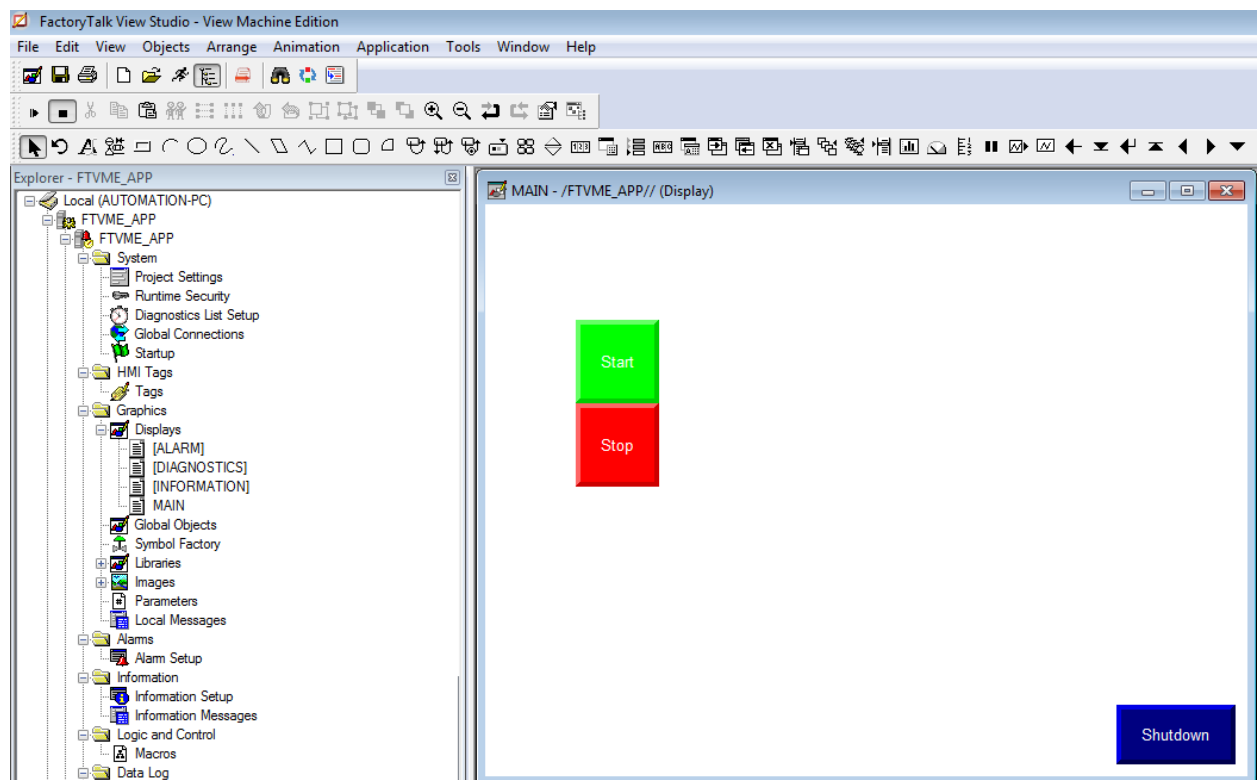
12. We can now test our newly created and tagged push button. From the toolbar of FactoryTalk View studio, click the “Test Display” button and then click on your start push button. You should notice an LED in your controller demo labeled “DO0” has turned on. If you don’t see any changed in your demo case, please seek help from one of the instructors. Click the “Stop” button, directly to the right of the test display button to stop testing your current display.



13. Use steps 2 through 12 from this lab to create another push button with the following information:

- Caption (State0/State1) : Stop
- Color : (State0/State1) : Red
- Height / Width : 70
- Tag : {[CMX]Stop}

Once you have created this push button arrange the new stop button so that it lines up vertically with the start button. Click the “test display” button and make sure you can now turn on and off the “DO0” LED with the start and stop buttons on your main display. If you have gotten this to work you are finished with this lab. If this does not work, please seek the help of one of the instructors. Your new “MAIN” display should look like the image below:



14. . Click the “Stop” button, directly to the right of the test display button to stop testing your current display.

Lab 7 Review

1. When referring to {[CMX]Start}, what does CMX stand for? What is start?
2. If you are in the browse tag window and can't see the tag you are looking for, what are the steps you would take to refresh the folder?
3. How can you test the current display you have open to make sure your PLC / PanelView connection is up and running?

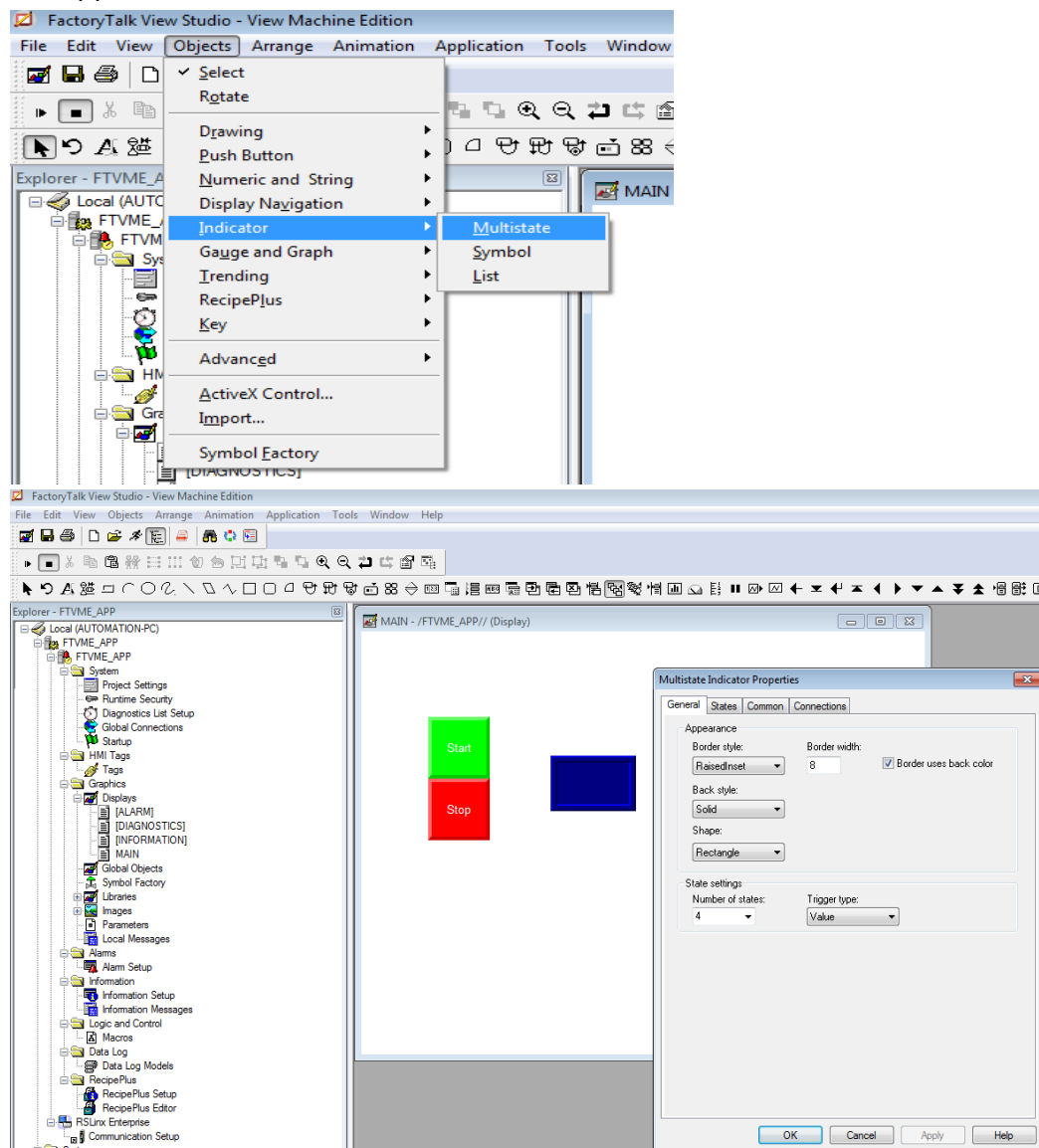


This completes lab 7. Please wait for instructions to proceed further.

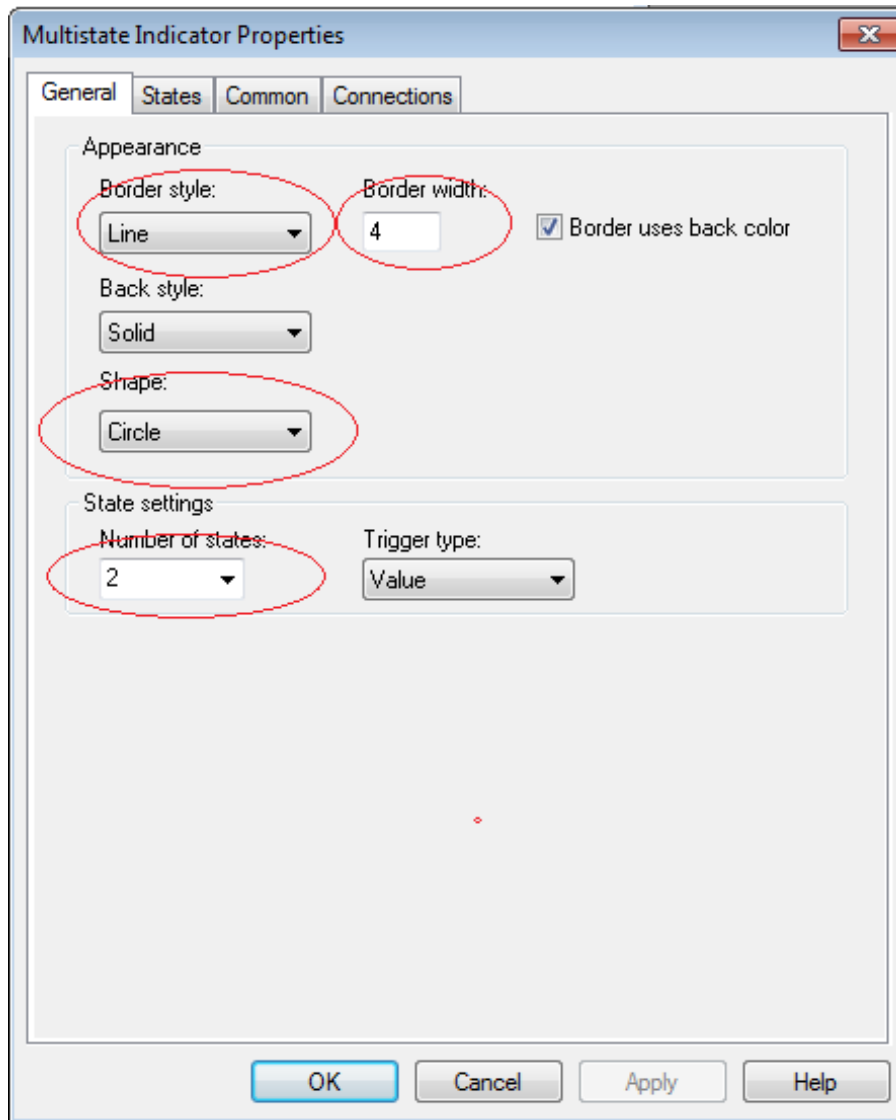
Lab 8: Creating an Indicator

Objective: To familiarize the user with the many uses of an indicator on a FactoryTalk View Studio display. This will be accomplished through the step-by-step creation of an indicator displaying the run condition of a motor, as determined by the push buttons created in the previous lab. At the end of this lab, the user will be familiar with the applications of an indicator, as well as their creation.

1. To add an indicator to your MAIN display, select “Objects→Indicator→Multistate”. You will notice your cursor change. Drag a small square with your mouse on the main display and once you release the mouse button, the “Multistate Indicator Properties” window will appear.

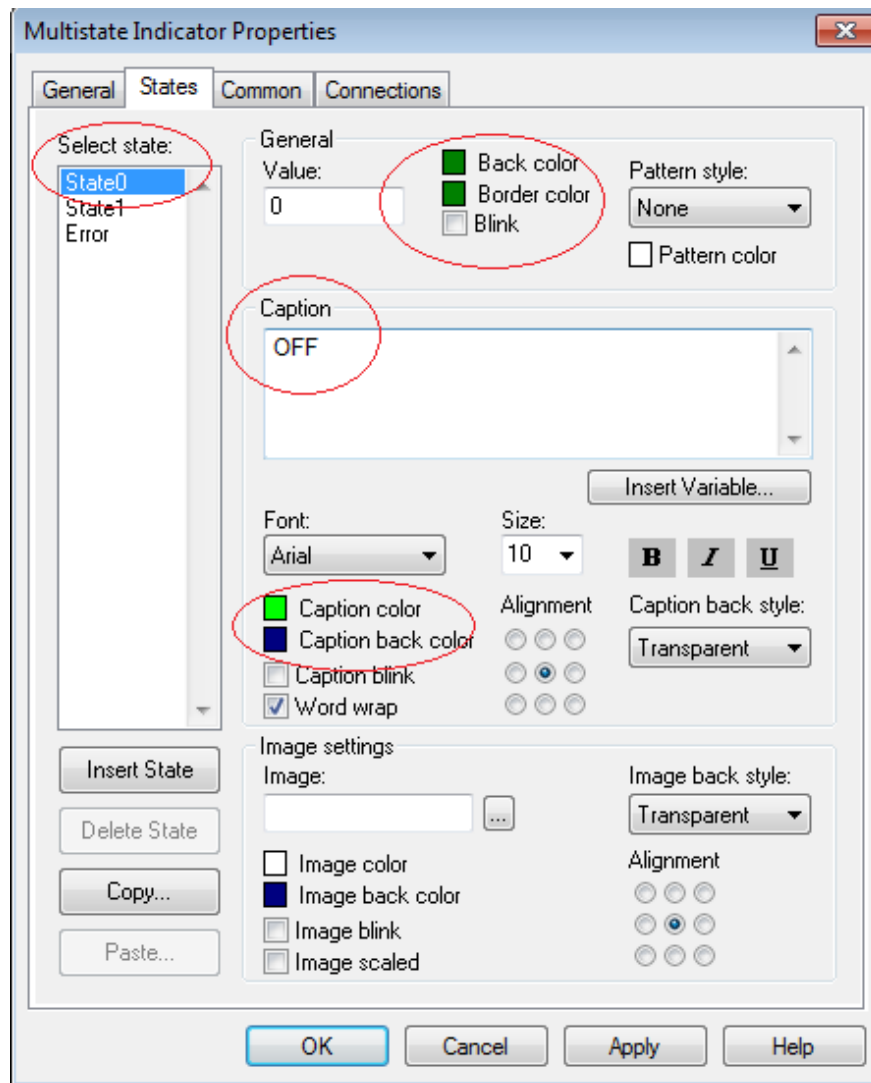


- From within the properties window in step 1, the default tab will be the “General” tab. From here, change the “Number of States” to 2 as shown below. Change the “Border Style to “line”, change the “border width” to 4 and change the “shape” to circle. Once this is done, click over to the “states” tab. We only need to states because this will be just an on/off indication. Your “General” tab should look like the image below



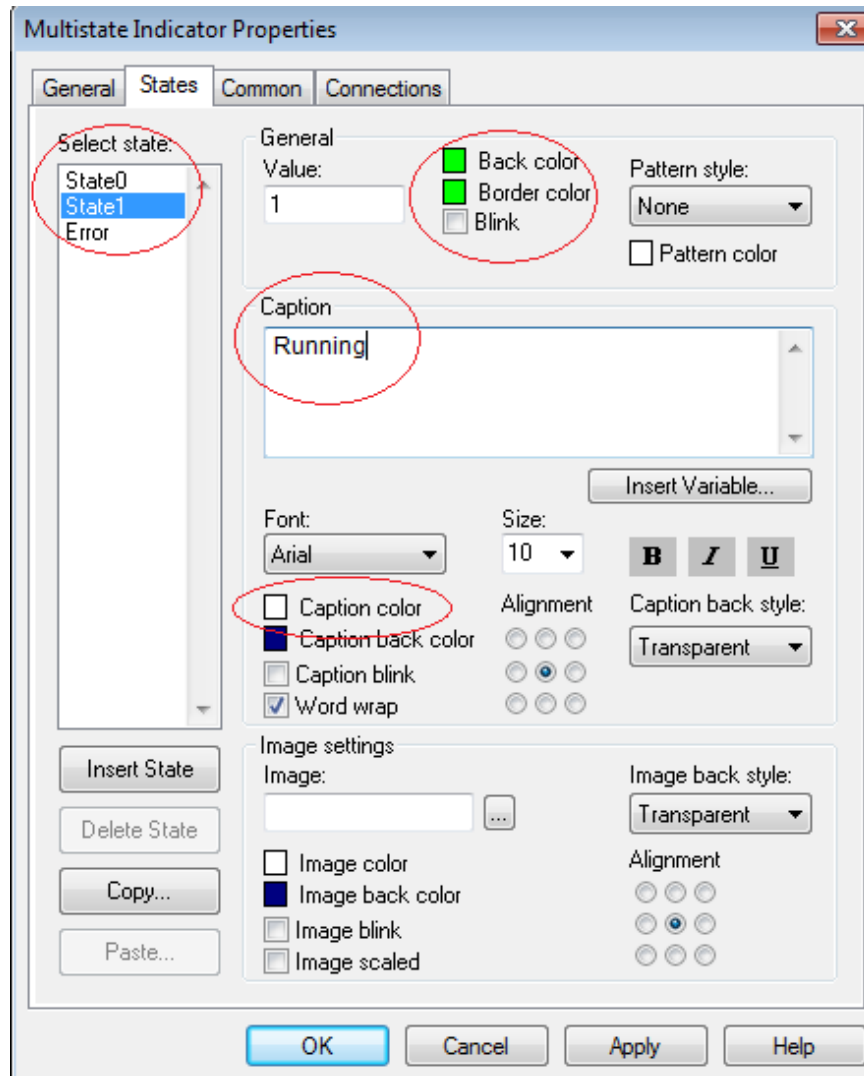
3. From the “States” tab of the indicator properties window, enter the following information for “State0”
- Back / Border Color: Dark Green.
 - Caption: OFF.
 - Caption Color: Light Green.

Your state0 configuration should look like the image below.

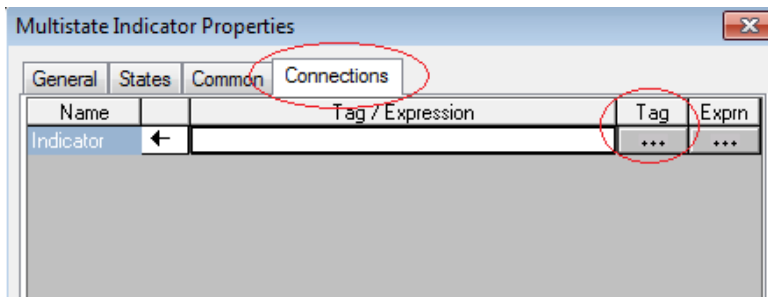


4. Once you have configured state0 from step 3, switch over to state 1 and enter the following information for “State1”:
- Back / Border Color: Light Green
 - Caption: Running
 - Caption Color: white

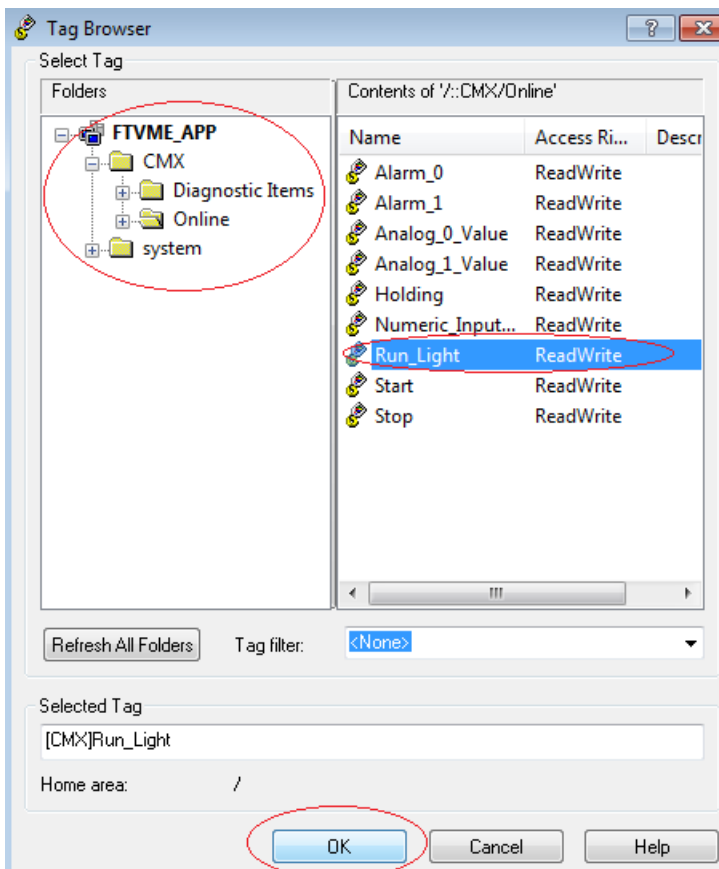
Your State1 configuration should look like the image below:



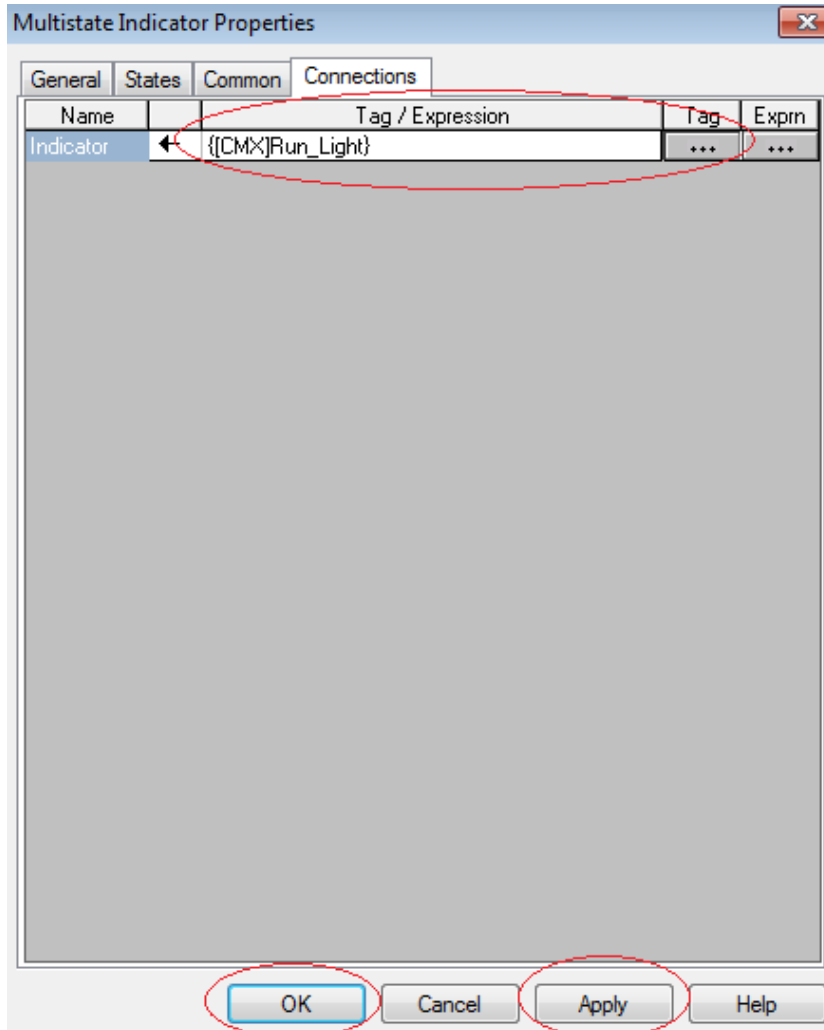
5. If you have completed steps 3 and 4 successfully, click over to the “Connections” tab of the properties window and select the browse button as shown in the image below.



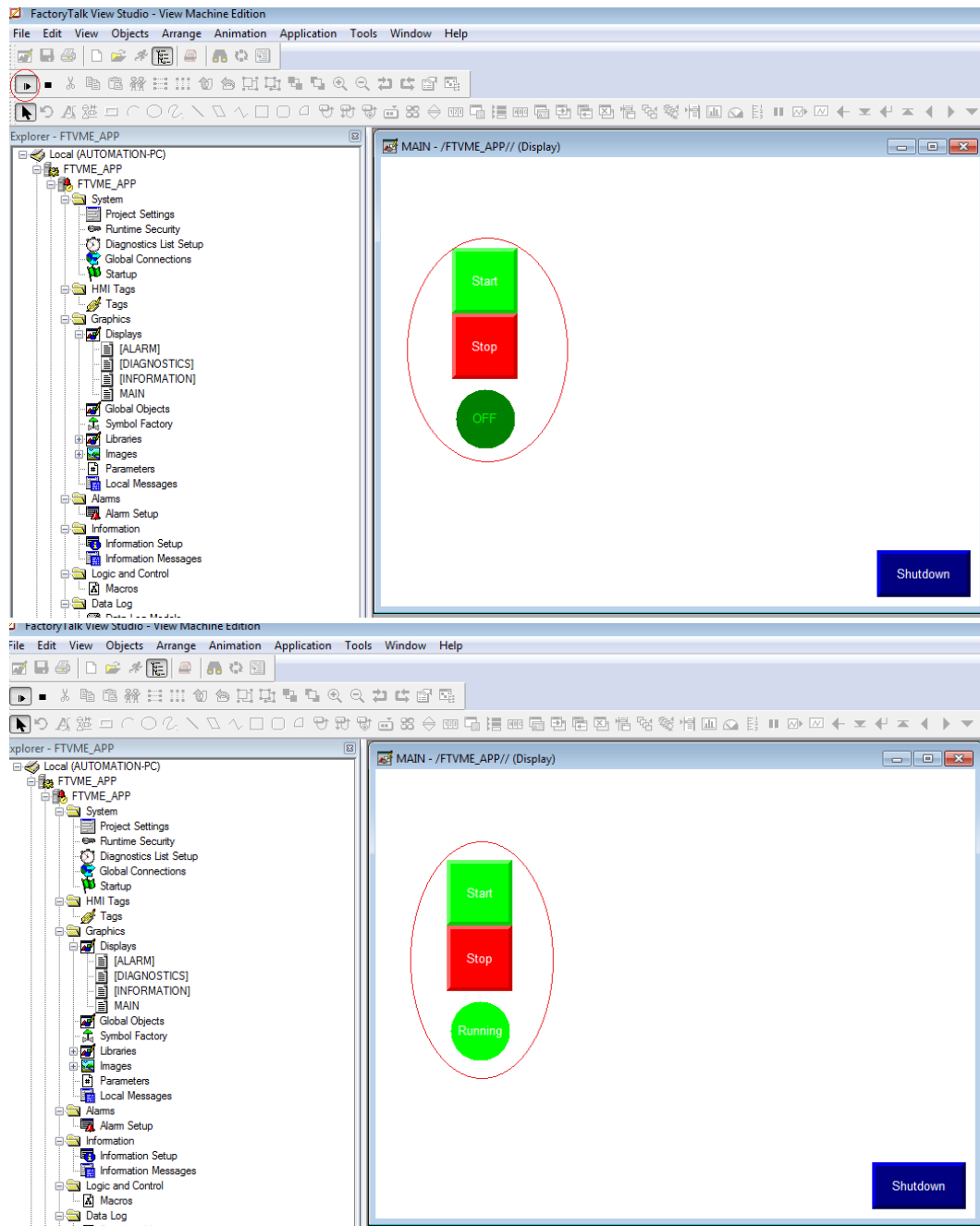
6. Browse to the CMX/Online folder which should already be open, select the Run_Light tag and then press the “OK” button at the bottom of the tag browser window.



7. After clicking “OK” from step 6, you should see the tag {[CMX]Run_Light} in the “Tag / Expression” field of the properties window. Click the “Apply” button followed by the “OK” button which will apply your changes and close the properties window.



8. Align your newly created and configured indicator vertically with the start and stop push buttons from the previous lab as show in the image below. Test your display by clicking the “Test Display” and make sure your indicator changes states as you press start / stop. If this does not work, please seek the help of one of the instructors. If your lab is working properly, click the stop button located directly to the right of the test display button.



Lab 8 review

1. In the Multistate properties window, what tab lets you change the number of states?
2. Can you place a different image on different states?
3. How many different states are possible?

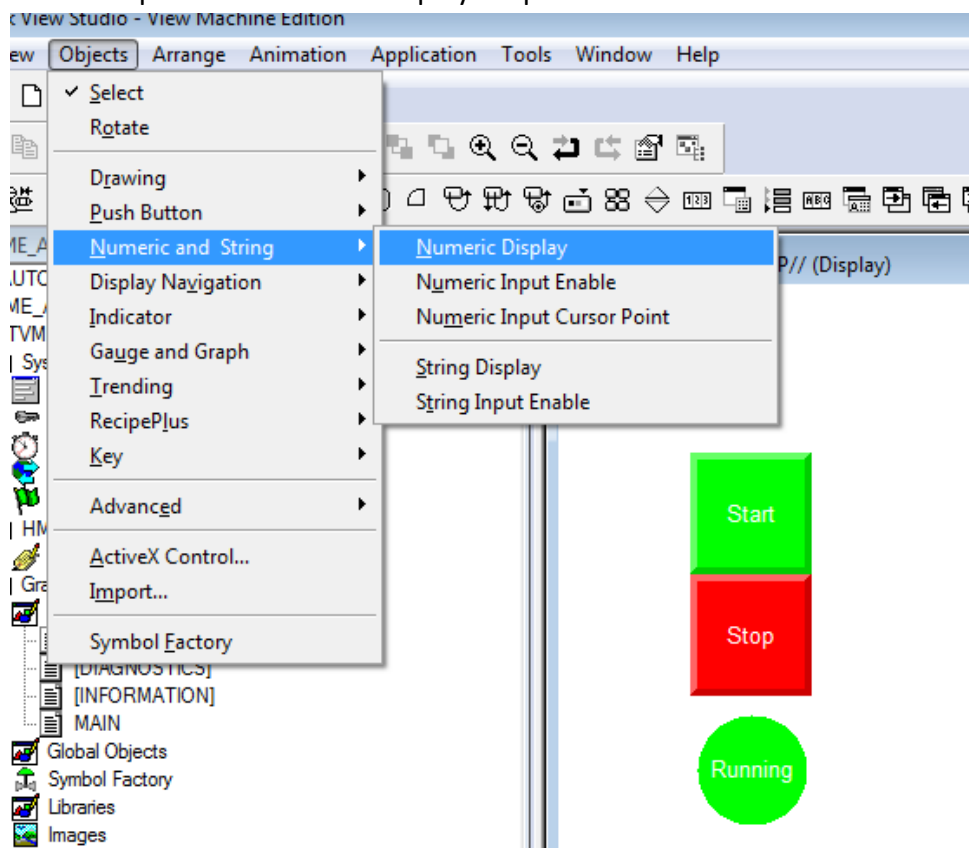


This completes lab 8. Please wait for instructions to proceed further.

Lab 9: Numeric Data Display

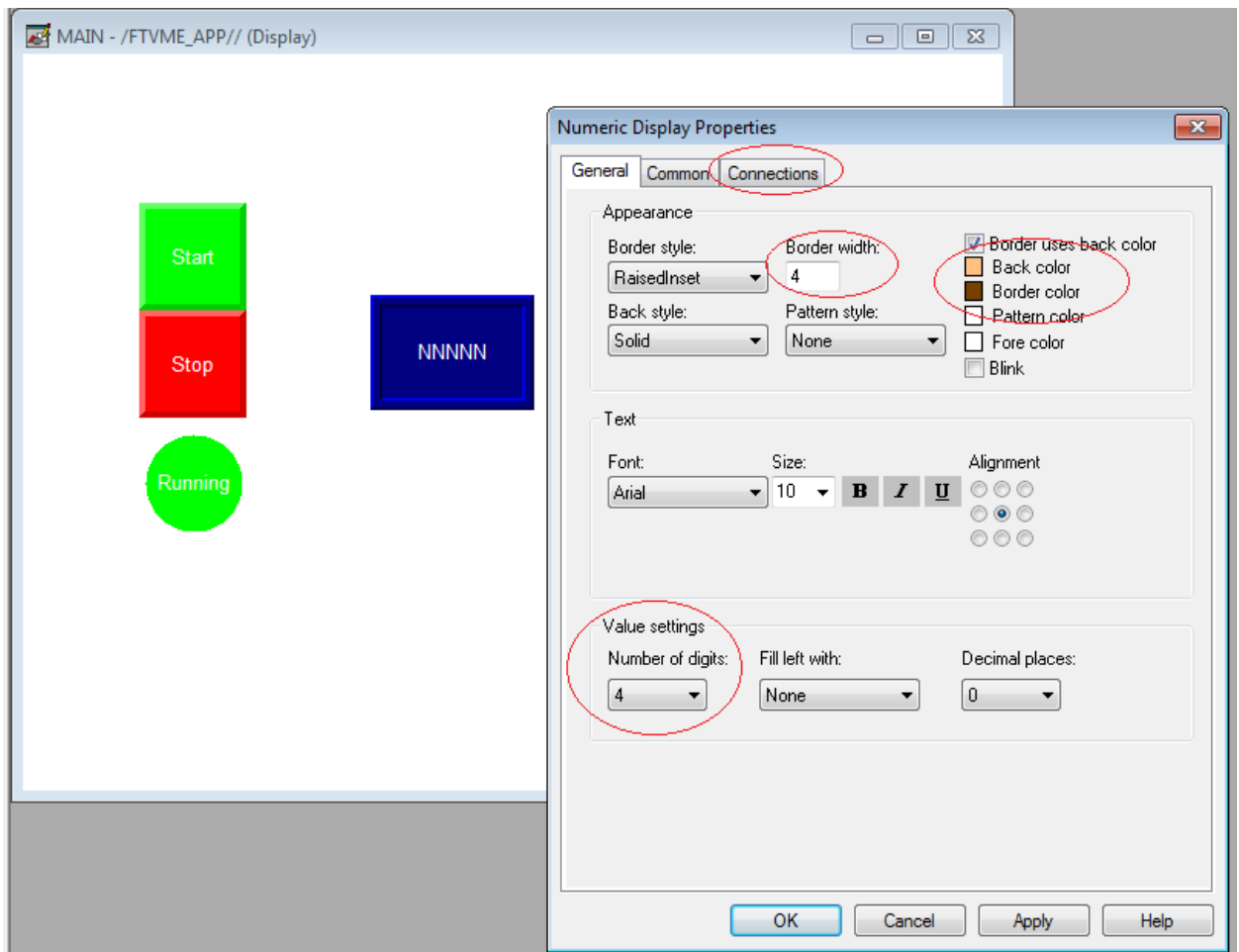
Objective: To introduce the user to the basic manipulation of a numeric data display object on a PanelView Plus display. To meet this goal, this exercise will cover step-by-step, creation, configuration and implementation of the numeric display. By the end of this lab, the user should be well-versed in displaying these data types, as well as determining when its use is advantageous.

1. The first thing we need to do to add a numeric data display, is to find it in the toolbar and draw one onto the screen. Start by going to “Objects→Numeric and String→Numeric Display”. You will notice that your cursor changes, so you can now draw a basic square on the screen by clicking and holding the mouse, dragging it across the display until you have made a small square and then releasing the mouse button. This will open the “Numeric Display Properties” window.

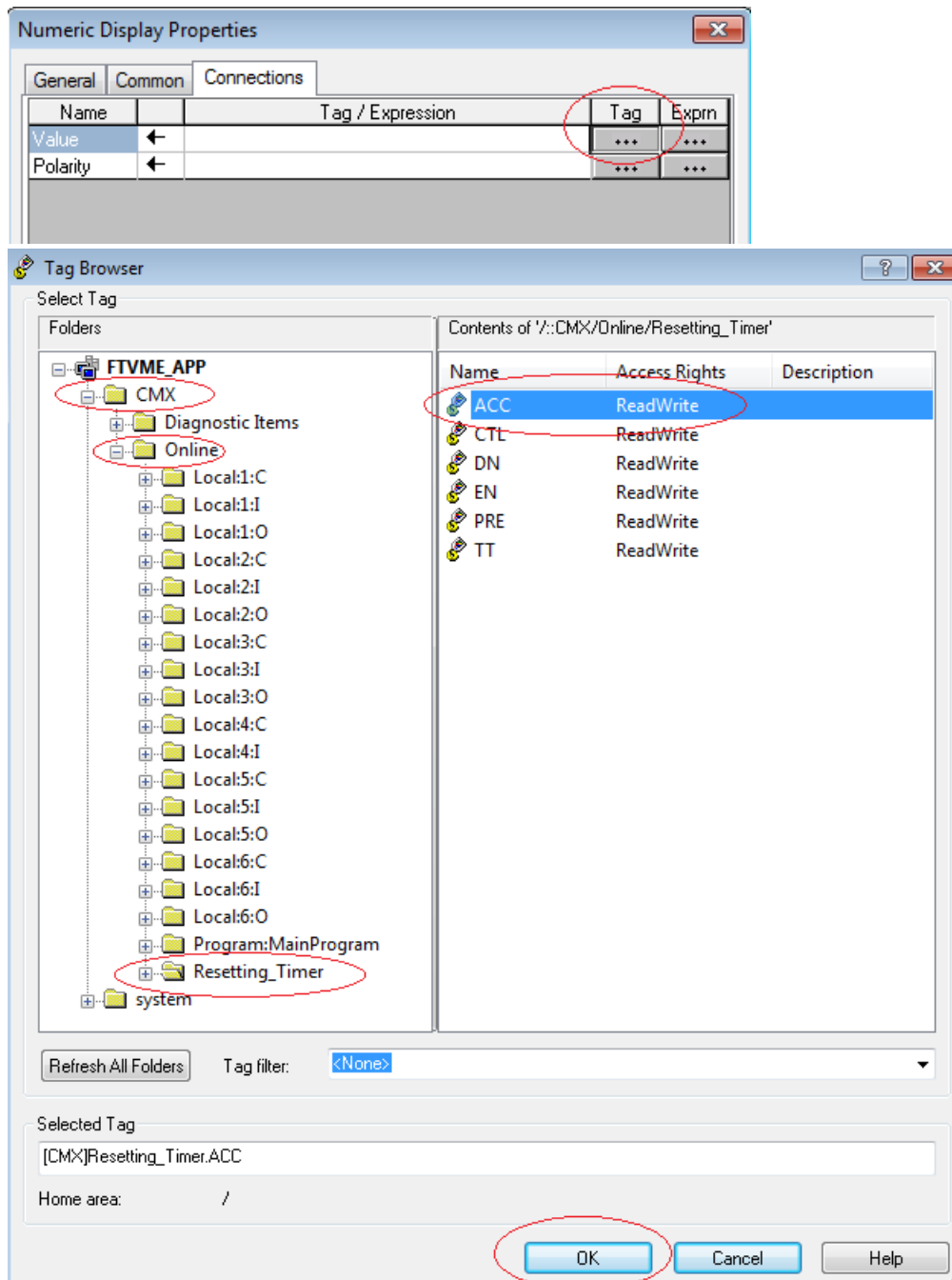


2. Once the numeric display properties window is open, we can start to configure our numeric display. From the general tab of the properties window, edit the following information:
- Border Width: 4
 - Back Color: Tan
 - Border Color: Brown
 - Number of Visible Digits: 4

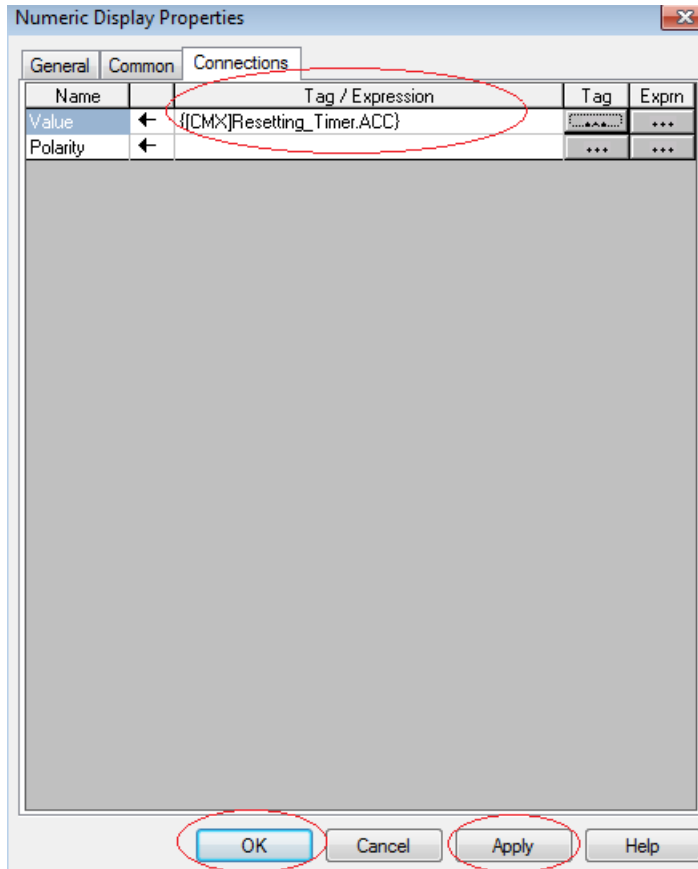
Your general tab should look like the following image. Once you have made these changes, click over to the “Connections” tab of the properties window.



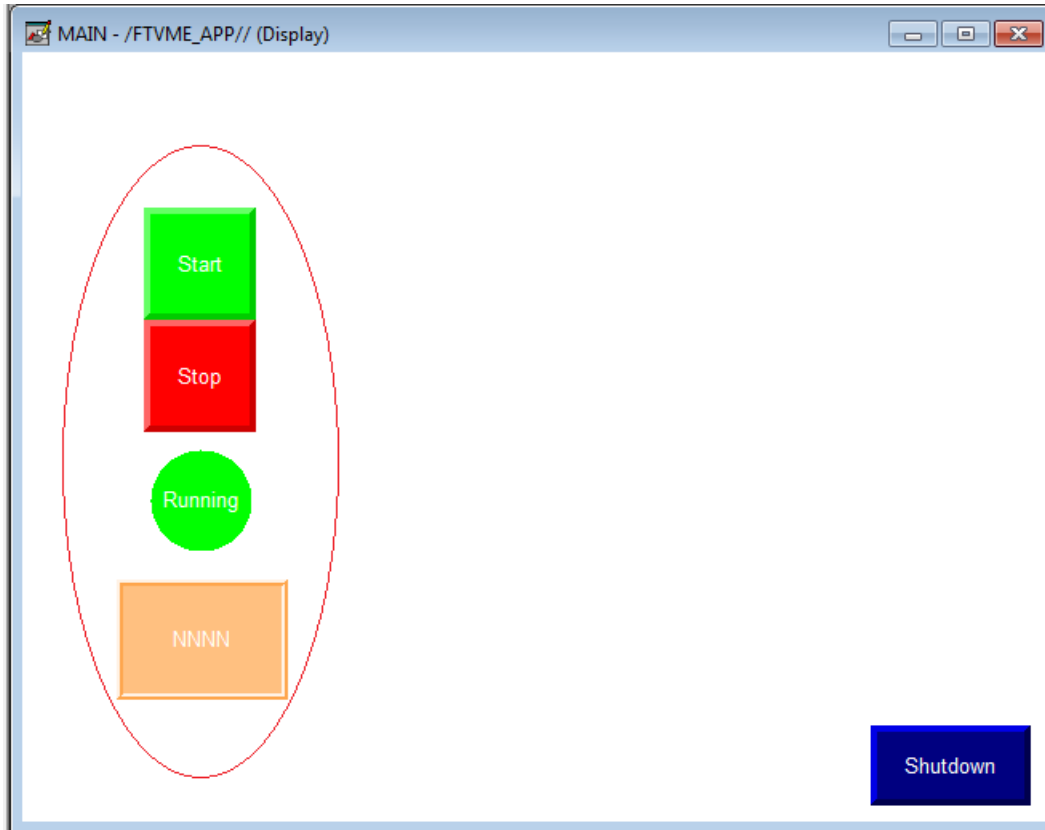
- From the connection tab, click the browse button to open up the tag browser. Browse for the tag “Resetting_Timer.ACC” which is the accumulated value of a timer in our PLC program. You can find this specific tag by following the path “CMX→Online→Resetting_Timer→ACC”. Once you have this tag highlighted, click the “OK” button at the bottom of the tag browser.



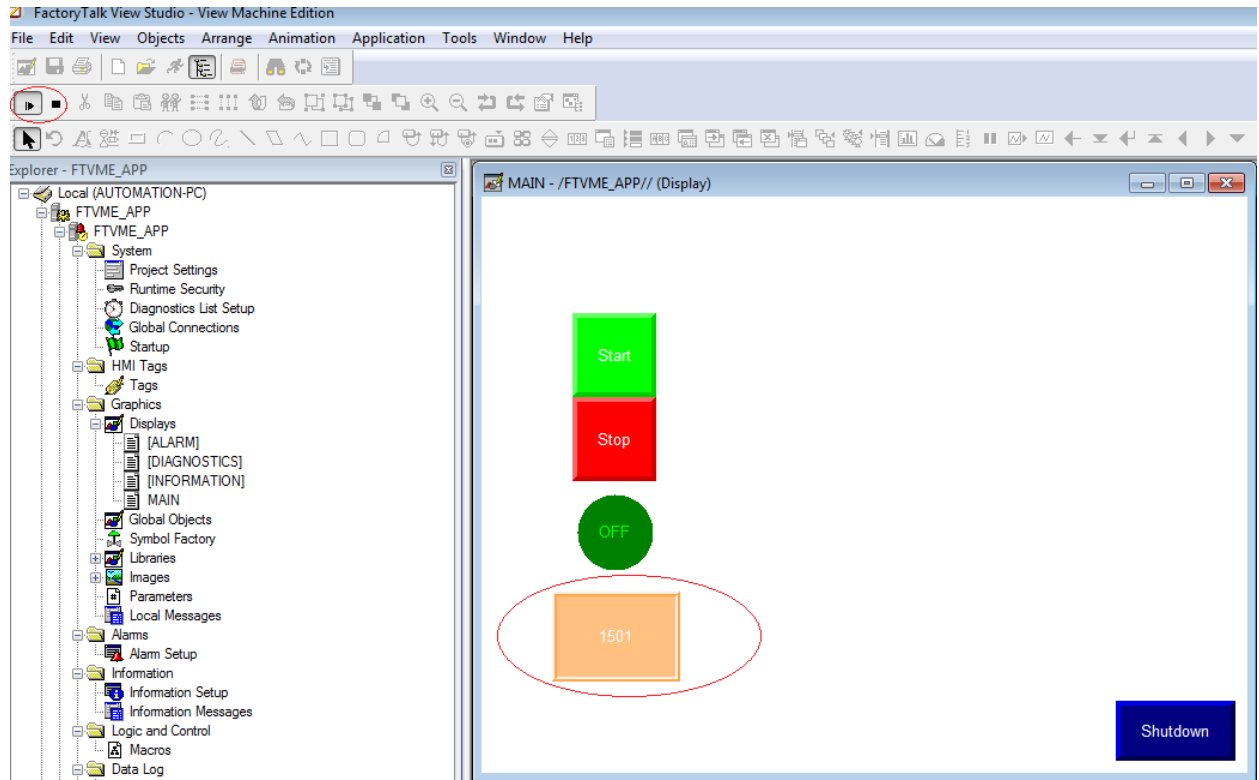
4. After clicking the “OK” button from step 3, you should see the tag “{[CMX]Resetting_Timer.ACC}” within the tag / expression field of the numeric display properties window. If your connection tab does not look like the image below, please seek the help of one of the instructors. Once you have this tag in place, click the “Apply” followed by the “OK” buttons at the bottom of the numeric display properties window.



5. You should now be back at your main display with the newly created numeric display. Drag the numeric display so that it lines up vertically with the start, stop and running indicator. Your MAIN display should look like the following image.



6. Test your display by clicking the “test display” button located in the upper left corner of the toolbar. Make sure you are seeing valid data on your newly created numeric display, as it should be displaying the accumulated value of a timer running in a loop within the PLC program. If you see the value changing, press the stop button next to the test button and you are finished with this lab.



Lab 9 Review

1. Within the properties window, which tab allows you to change the number of visible digits?
2. Where is the tag update rate located?
3. How would you display a decimal value to the hundredths place?

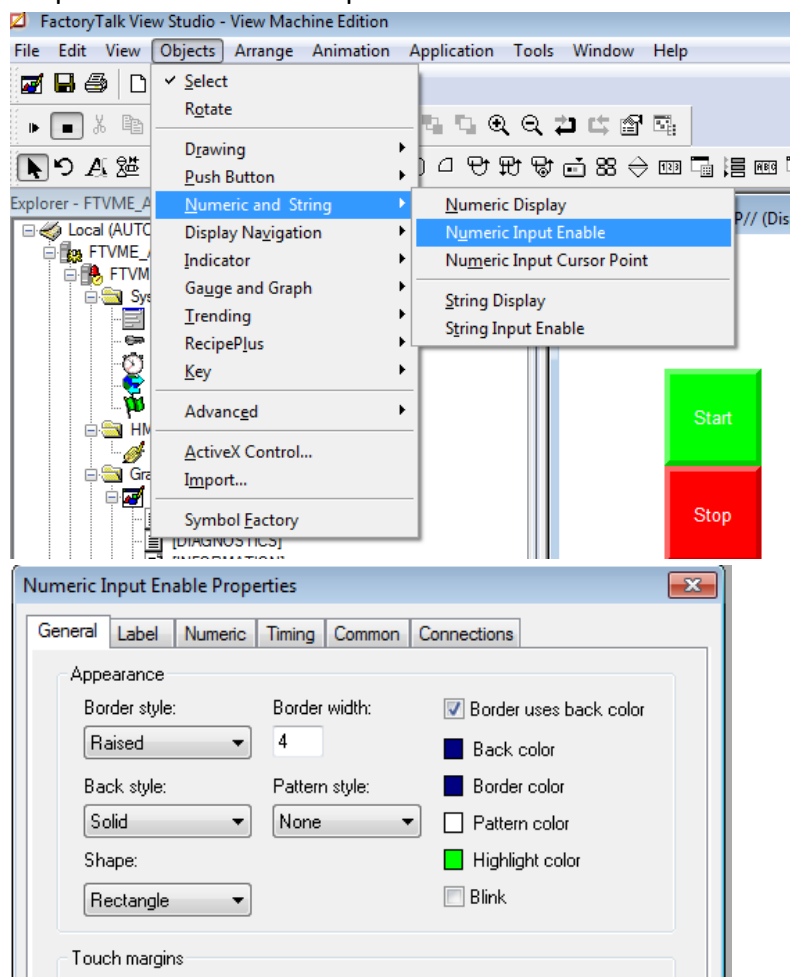


This completes lab 9. Please wait for instructions to proceed further.

Lab 10: Numeric Input Enable

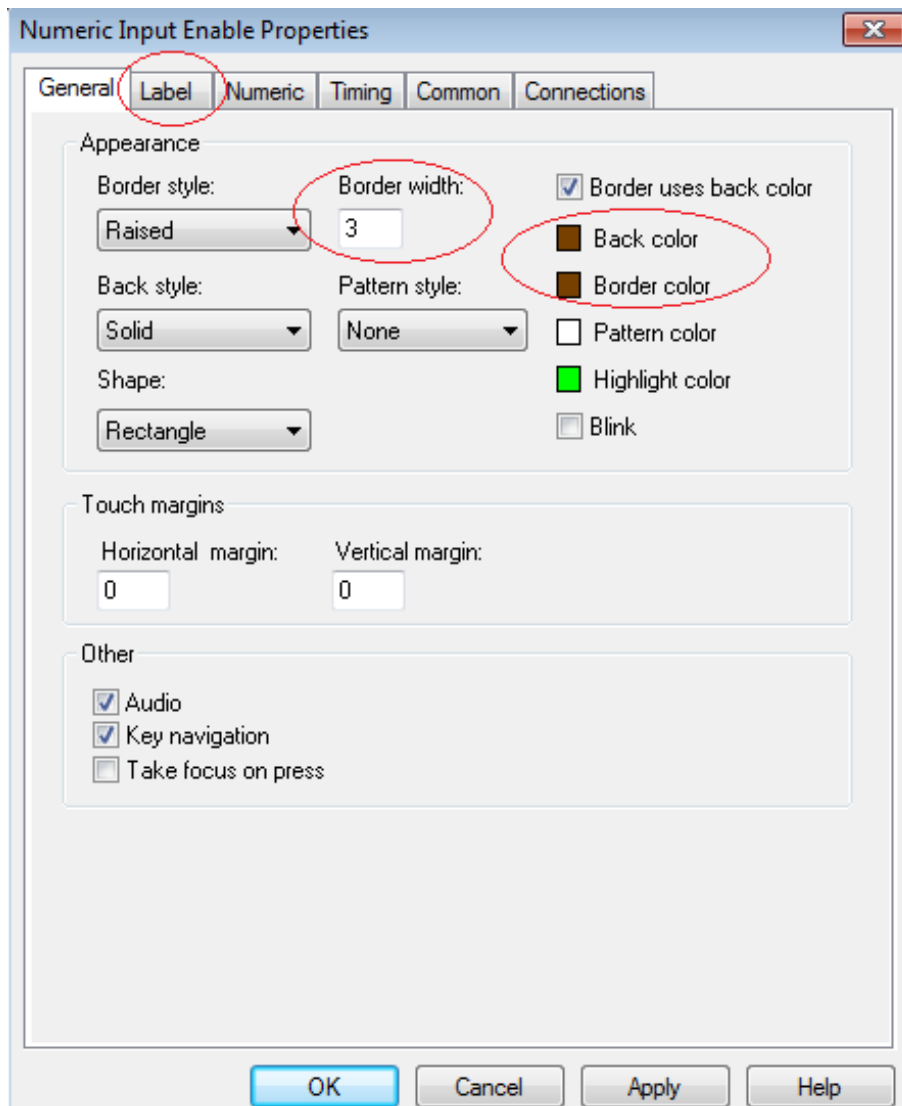
Objective: To familiarize the user with the properties of the numeric input enable through creation and configuration of the object. By the end of the lab, the user should be capable of easily creating a numeric input enable that can handle valuable operator defined information.

1. The first thing we need to do to create a numeric input enable is to find it in the tool bar and draw one on our main display. You can find the numeric input enable object at “Objects→Numeric and String→ Numeric Input Enable”. Once you have selected this, draw a basic square on the screen, after doing this the “Numeric Input Enable Properties” window will open.



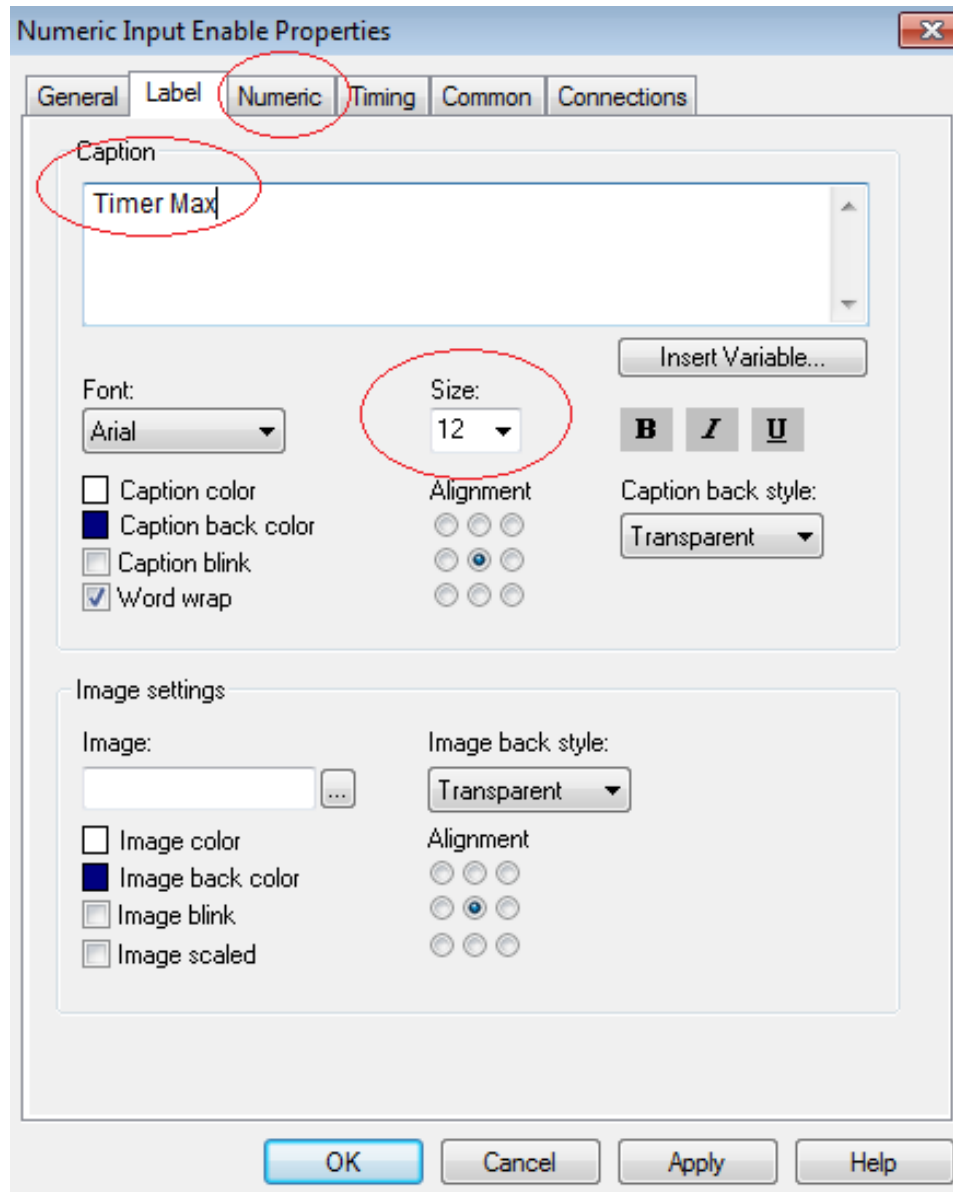
2. The default tab within the properties window will be the “General” tab. From this tab, enter in the following information:
 - Border Width: 3
 - Back / Border Color: Brown

Your general tab should look like the following image. Once you have made these changes successfully, click over to the “Label” Tab.



3. From the “Label” tab, enter in the following information:
- Caption: Timer Max
 - Font Size: 12

Your “Label” tab should look like the following image. If you have made these changes successfully, click over to the “Numeric” tab.



4. From the “Numeric” tab, enter in the following information:
- Numeric pop-up: Keypad
 - Minimum Value: 500
 - Maximum Value: 6000

Your numeric tab should look like the following image. If you have made these changes successfully, click over to the “Connections” tab.

The image shows a screenshot of the "Numeric Input Enable Properties" dialog box. The "Numeric" tab is selected, and the "Connections" sub-tab is highlighted. The "Pop-up settings" section shows "Numeric pop-up" set to "Keypad". The "Min/max range and ramp settings" section shows "Minimum value" set to 500 and "Maximum value" set to 6000. The "Decimal point settings" section shows "Decimal point" set to "Implicit" and "Fixed position output" set to "Stripped Value".

Numeric Input Enable Properties

General | Label | **Numeric** | Timing | Common | **Connections**

Pop-up settings

Numeric pop-up:
Keypad

Min/max range and ramp settings

Minimum value: 500 Maximum value: 6000 Ramp value: 0

☐ Use variable minimum/maximum

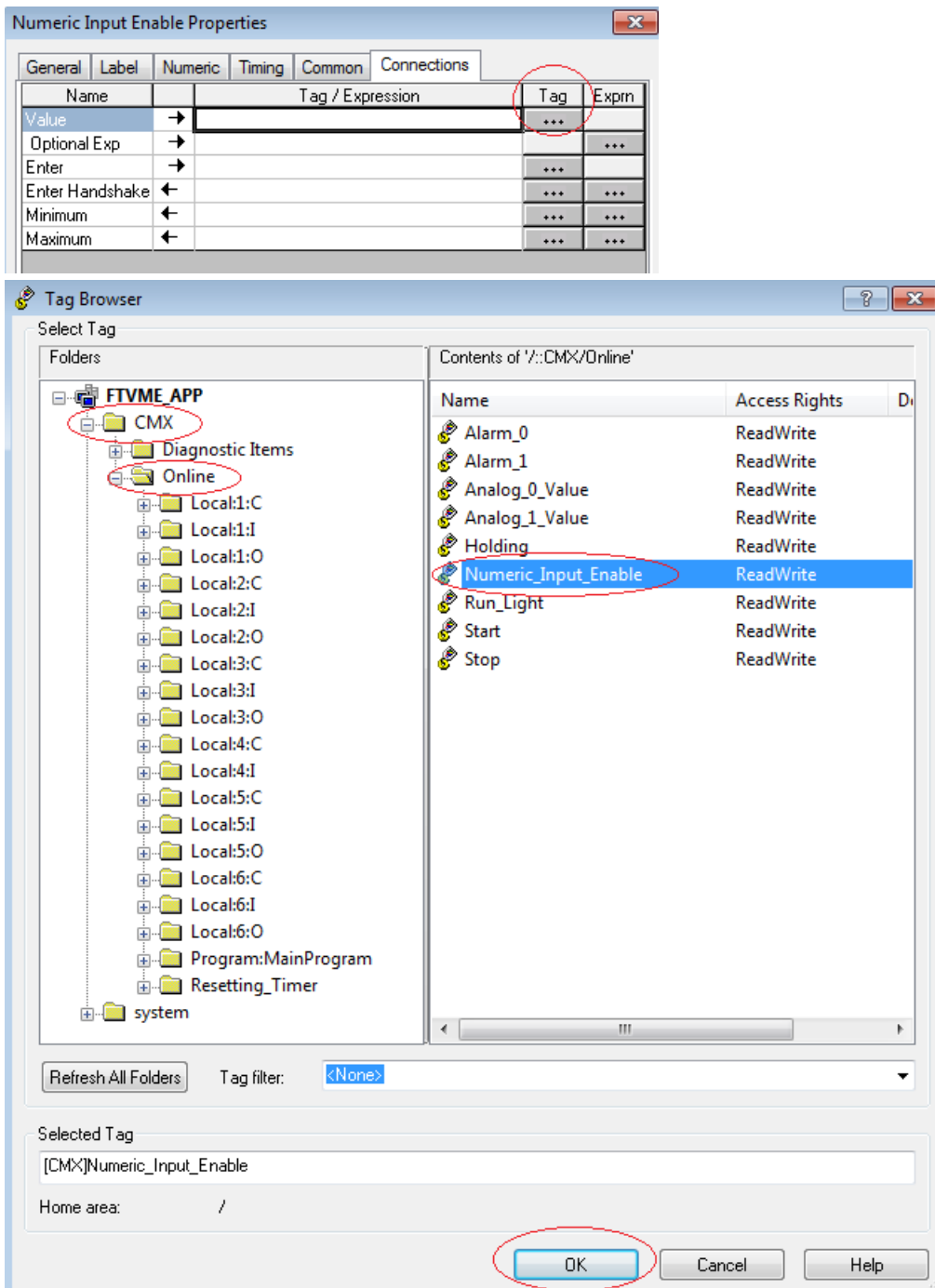
Decimal point settings

Decimal point: Implicit Fixed position output: Stripped Value

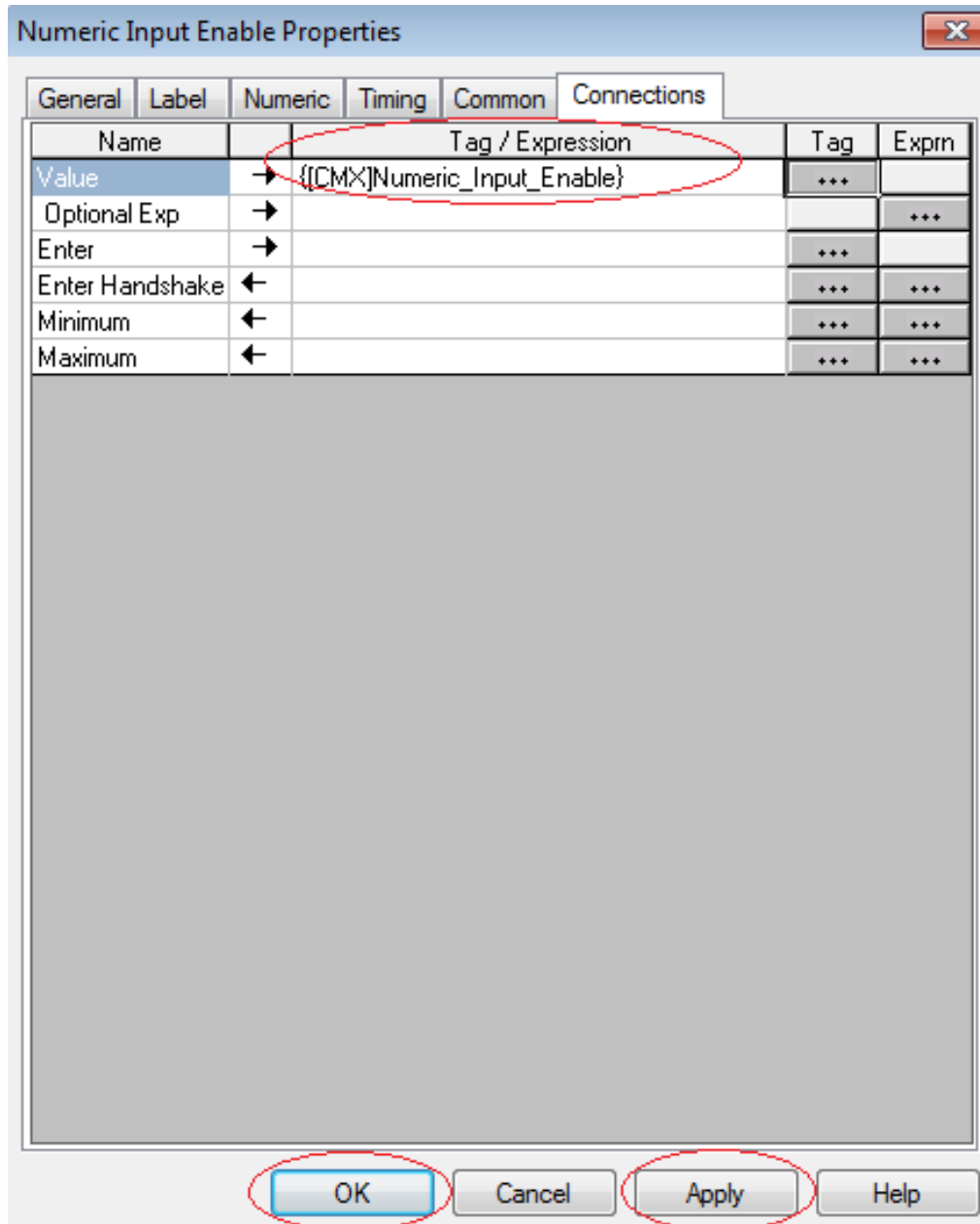
Digits after decimal: 0

OK Cancel Apply Help

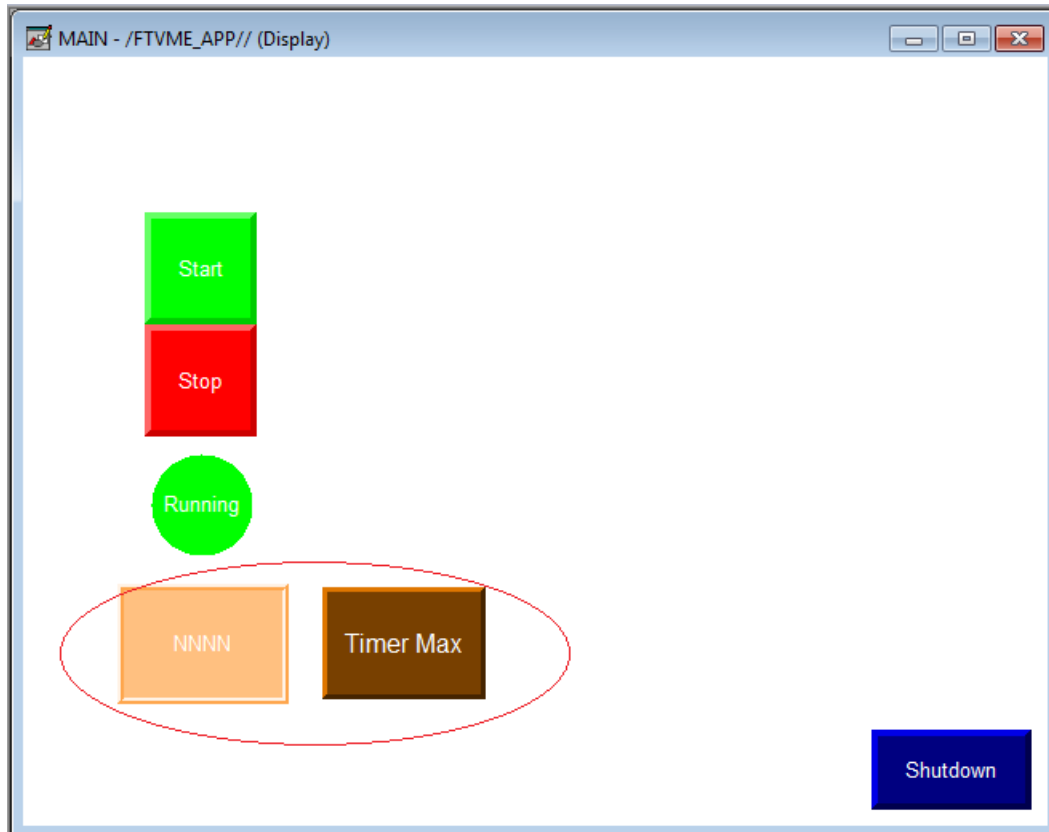
5. From the “Connections” tab, click the browse tag button as shown in the image below. Clicking the browse button will open the tag browser window so that we can look for a tag to link our object to. The tag we are looking for is the “Numeric_Input_Enable” tag that can be found at “CMX→Online→Numeric_Input_Enable”. Highlight this tag and press the “OK” button at the bottom of the tag browser window. This will place the tag we selected into the tag / expression field of our numeric input enable properties window.



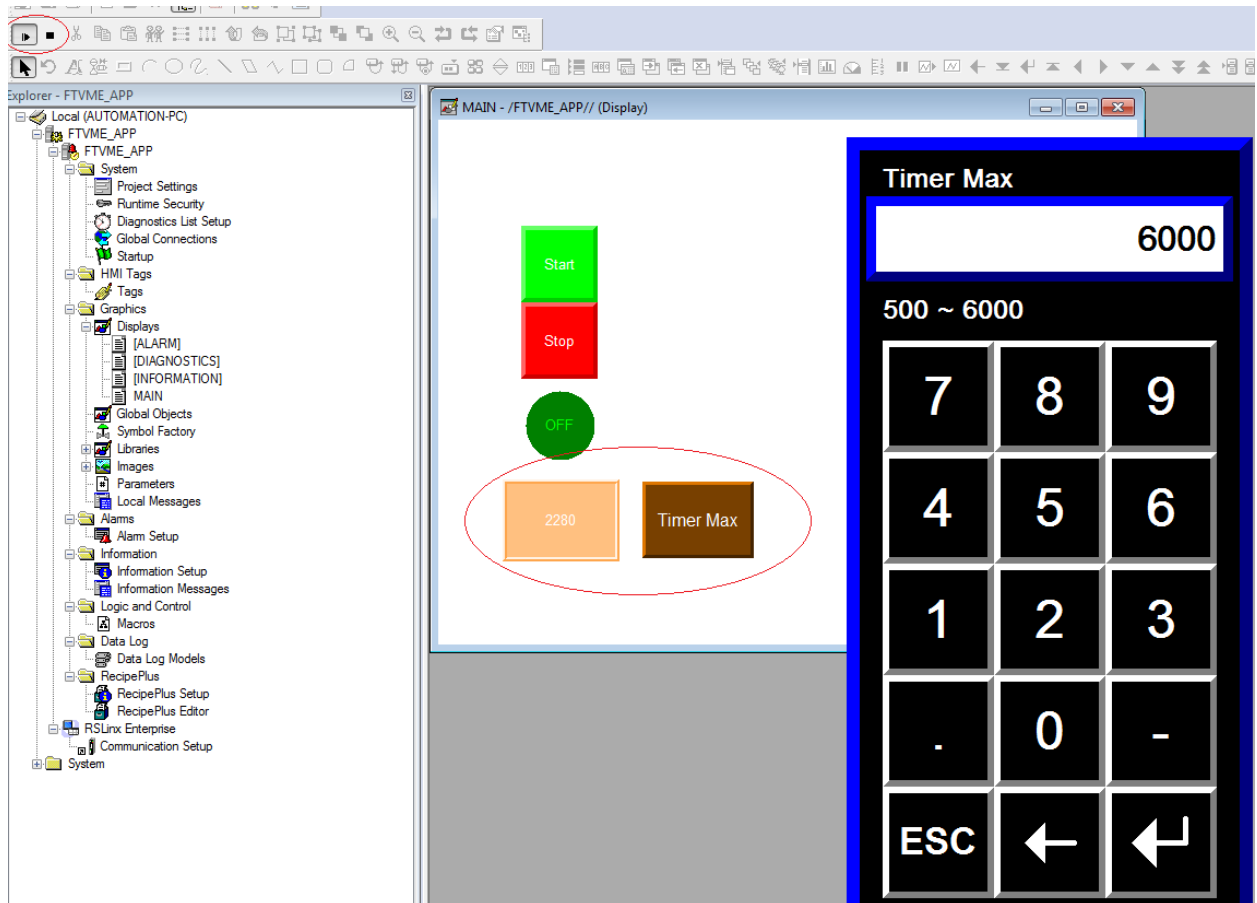
6. After clicking “OK” in step 5, You should see the tag “Numeric_Input_Enable” within the tag / expression field of the numeric input enable properties window. If your properties window does not look like the image below, please seek the help of one of the instructors. If your properties window is correct, click the “Apply” followed by the “OK” buttons at the bottom of the numeric input properties window.



7. After clicking “Apply” and “OK” from step 6, you will be brought back to your MAIN display with your newly created numeric input enable object. Line this object up just to the right of the numeric display object from the previous lab, as shown in the image below



8. Test your display and numeric input enable. Click the “test display” button and try clicking on your new numeric input enable object. You should be able to change the value of the timer preset running in the PLC within a range of 500 to 6000. After changing values, make sure your numeric display reflects the value you set in the numeric input enable. For example, if you enter a value of 5000, your timer and numeric display should only go up to 5000 before resetting.



9. After confirming that your application works up until this point, click the “stop” button located to the right of the test display button and you will have completed this lab. If your numeric input enable doesn’t work, please seek the help of one of the instructors.

Lab 10 Review:

1. In the numeric input enable properties window, what tab can you set the minimum and maximum values?
2. If you set the numeric pop-up to scratch pad, what must be attached to the PanelView in order to enter values?
3. What happens if you enter the value of 450 on the keypad of the numeric input enable?

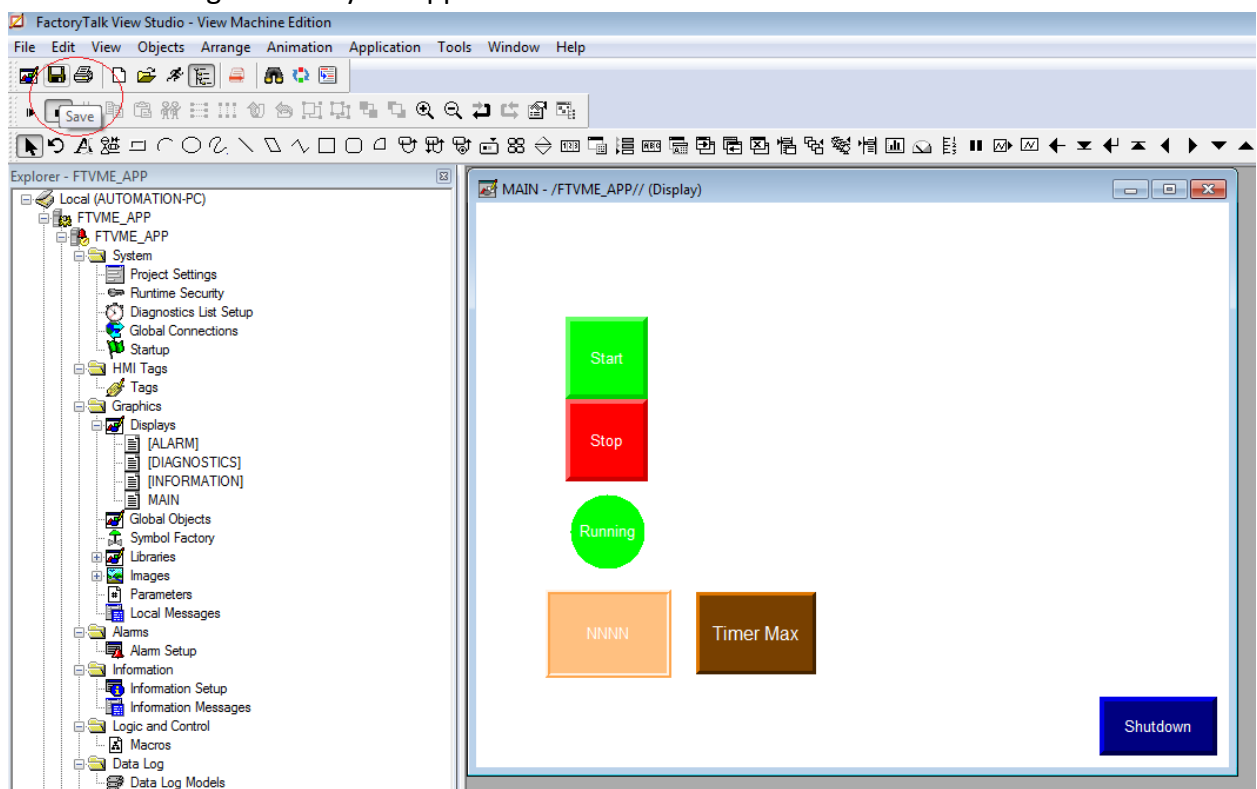


This completes lab 10. Please wait for instructions to proceed further.

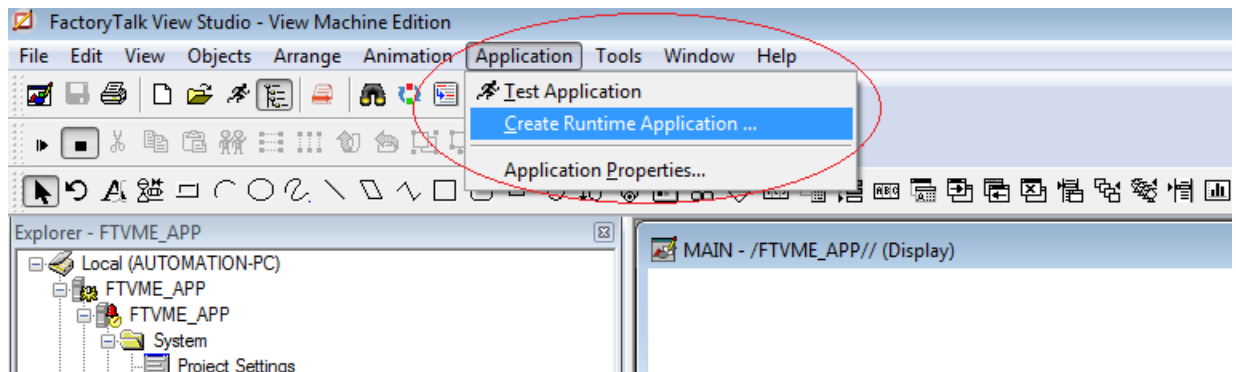
Lab 11: Saving / Downloading an application

Objective: To introduce the user to the concepts of saving and downloading PanelView Plus programs (.MER files) as well as basic PanelView Plus communications. By the end of this lab, the user will be able to create the runtime file on their local PC, as well as downloading that file to the PanelView.

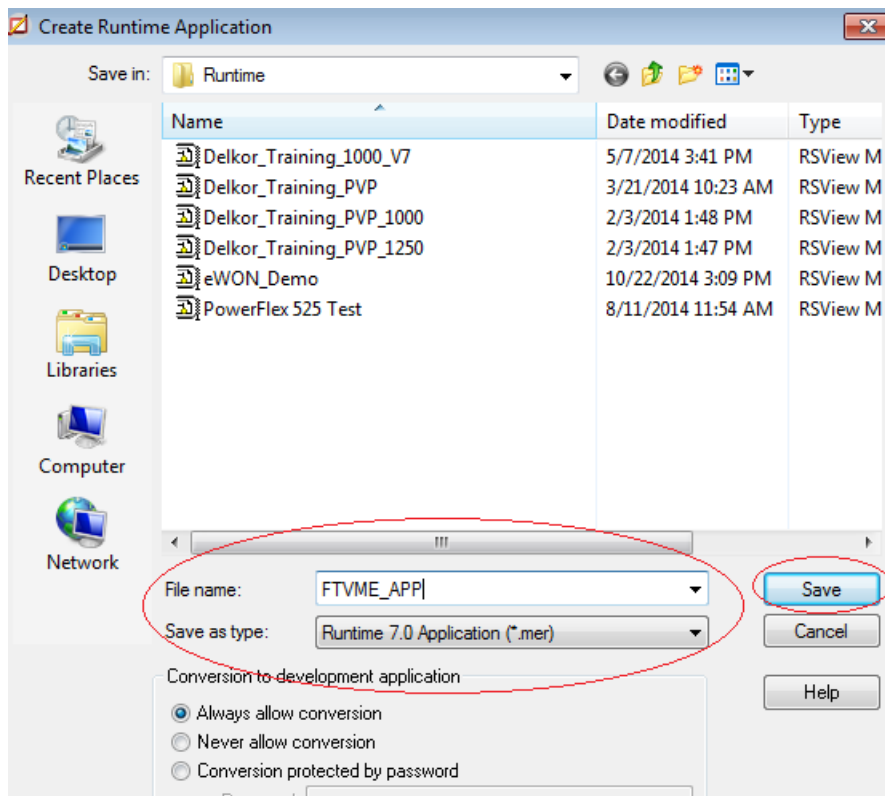
1. The first thing we need to do is save the work we have done within our application. There are a couple ways to save the application. You can save by going to “File→Save”, or you can simply click on the disk icon in the toolbar as shown in the image below. Click on the disk image to save your application



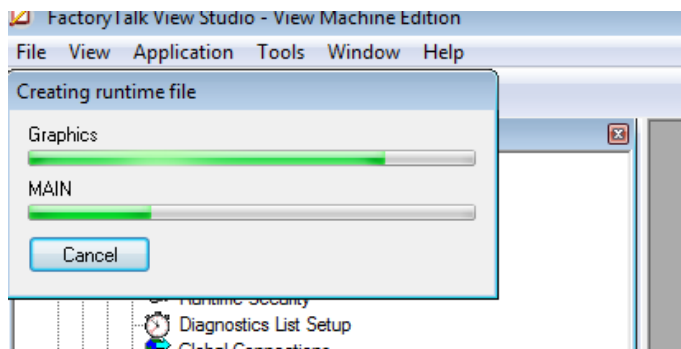
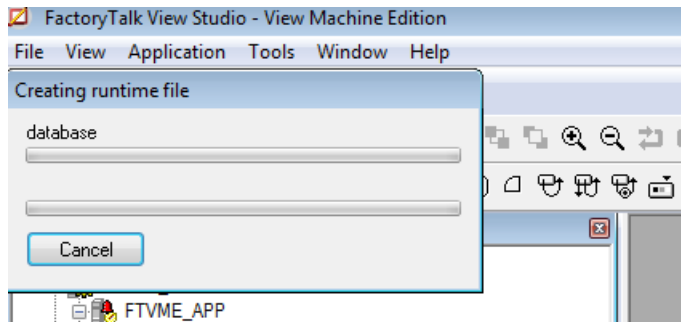
- The next step we need to take is to create the actual “.MER” file or “Runtime” file, that will be sent to the PanelView so that it can run on the terminal itself. Start by going to “Application→Create Runtime Application”.



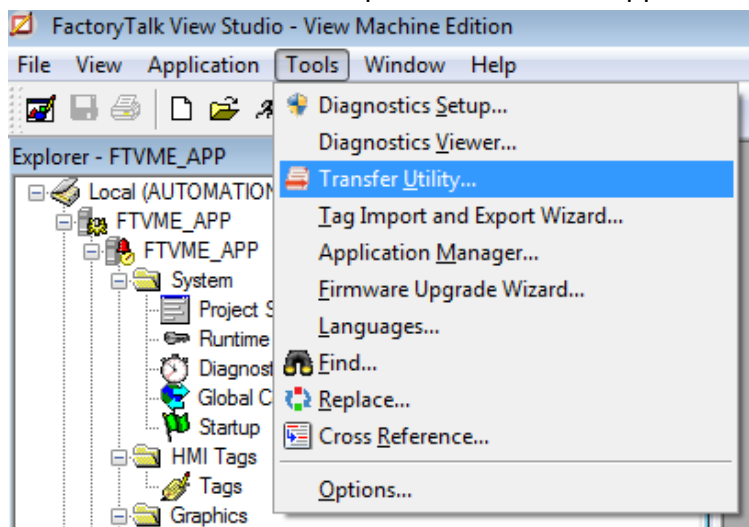
- Once you click “Create Runtime Application”, you will be prompted to name your application. In this case, we will leave the default name of “FTVME_APP” in the filename field. We will also need to specify the firmware revision in which to create this file. In this case we will leave it at the default of “Runtime 7.0 Application (*.mer)”. Make sure your create runtime application window looks like the image below. Once you have made the appropriate changes, click the “Save” button.



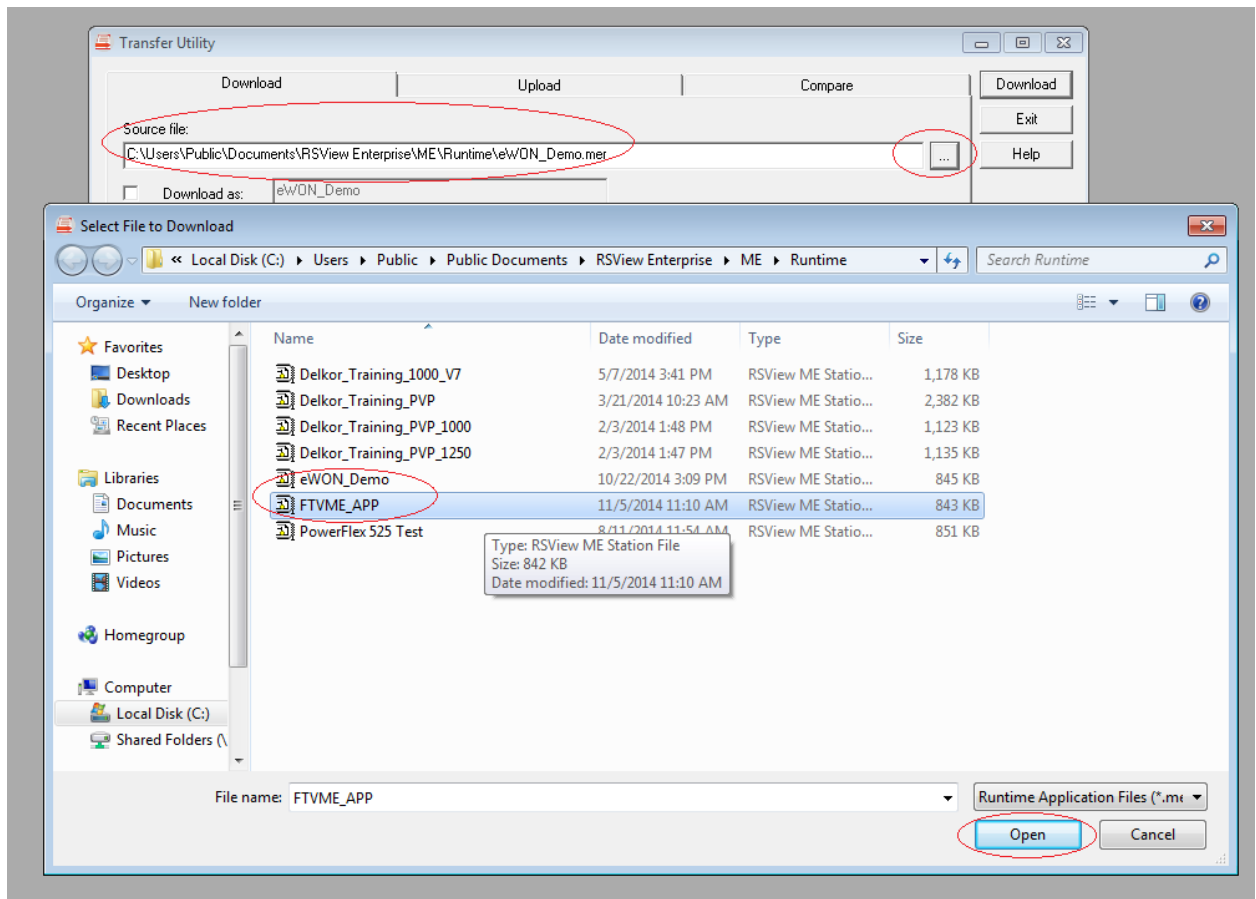
- Once you click “Save” in step 3, the creation process of the runtime file will begin. You will see a couple different loading bars and will need to wait until those are complete.



- Once the loading bars have completed, your runtime file will have been successfully created. The next step is to use a utility within FactoryTalk View Studio call the “Transfer Utility”. This utility can be found by going to “Tools→Transfer Utility”. The transfer utility is needed to download or upload a PanelView application to or from the actual terminal.



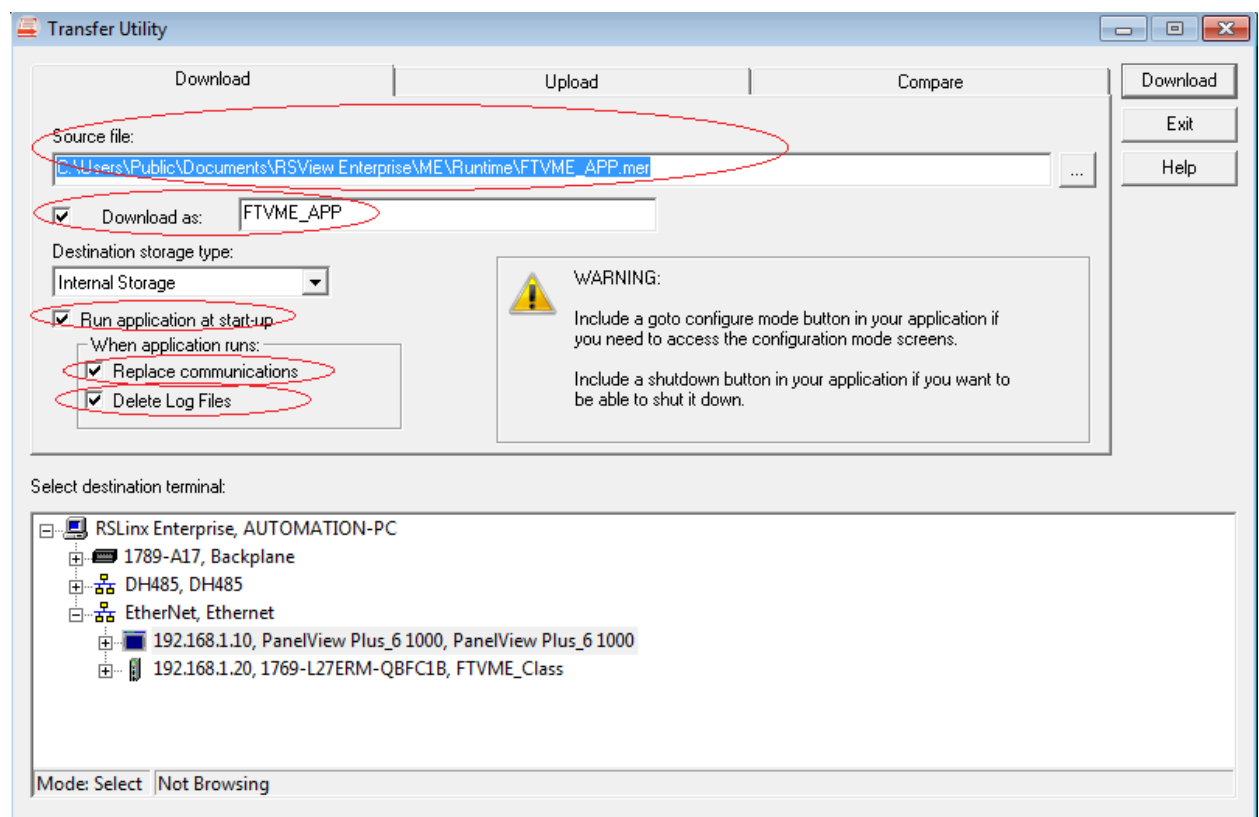
6. After clicking “Transfer Utility” from step 5, the transfer utility window will open. There are a couple pieces of information we are concerned with in this window. The first thing we need to do is make sure that we are downloading the correct .MER file to our PanelView. You will notice a “Source File” field near the top of the utility window. Our project file needs to be listed in this field. Start by clicking the “browse” button as shown below, and browse to the .MER file we created in step 4. Once you have located the .MER file, highlight the file and click the “Open” button at the bottom of the window.



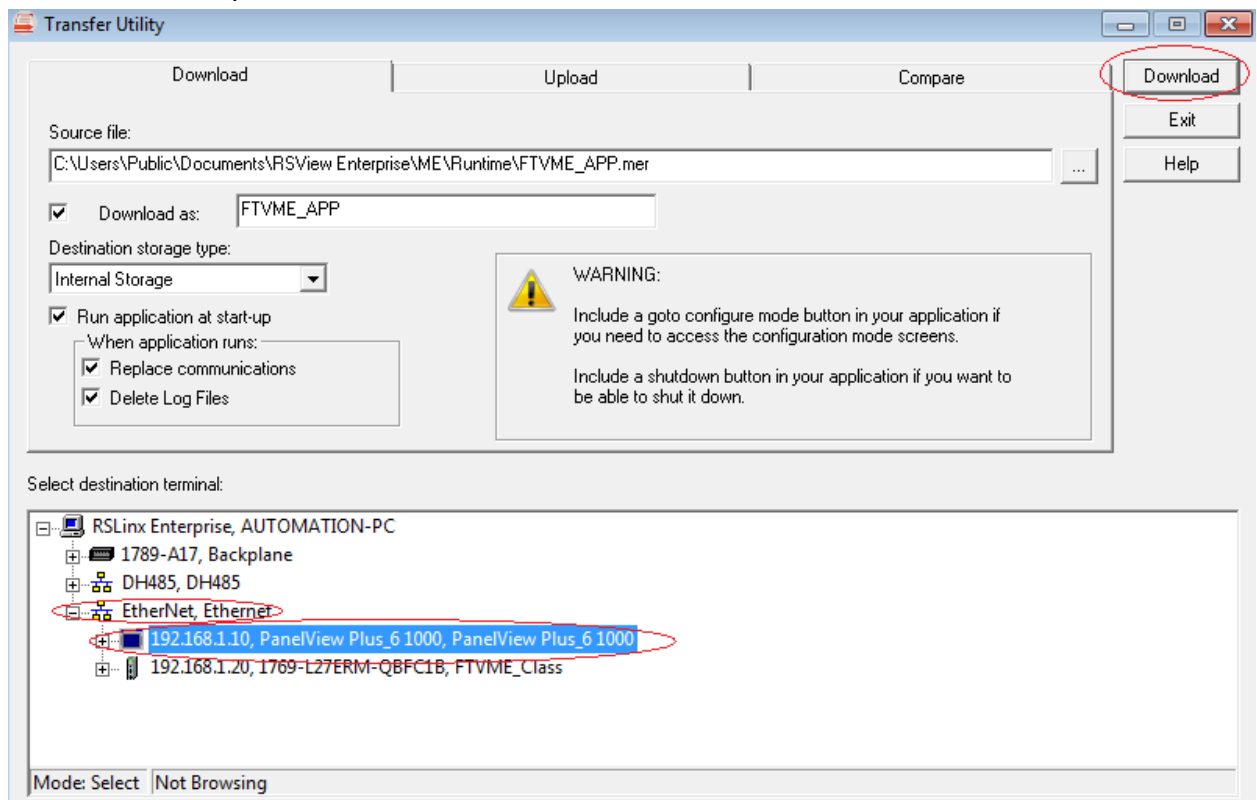
7. After clicking the “Open” button from step 6, you should now see the correct application name within the “Source File” window. The next step we need to take is specifying how the PanelView will behave upon startup. Start by checking the following check-boxes:

- Download as (This option will assure the correct .MER file is downloaded)
- Run Application at start-up (This option will make sure our .MER file runs at power-up)
- Replace Communications (This option will replace any existing communication settings in the PanelView with the ones we have configured in this course)
- Delete Log files (this option will delete any existing log files which is always a best practice)

Your transfer utility window should now look like the image below. If your utility window does not look like the image below, please seek the help of one of the instructors before proceeding further.



8. Before the download will begin, we now need to specify which terminal to download our .MER file to. You will notice at the bottom of the transfer utility an RSLinx Enterprise window which looks similar to RSLinx Classic. Start by clicking the “+” next to the EtherNet,Ehernet driver if it is not already expanded. Next, highlight the PanelView plus listed as 192.168.1.10 as shown below in the image. Once you highlight the PanelView in the RSLinx window, click the “Download” button at the top right corner of the transfer utility window.



-
- The image displays two screenshots of the Rockwell Transfer Utility software interface.
- Top Screenshot:** The 'Download' tab is active. The 'Source file' is 'C:\Users\Public\Documents\RSView Enterprise\ME\Runtime\FTVME_APP.mer'. The 'Download as' field is 'FTVME_APP'. The 'Destination storage type' is 'Internal Storage'. A 'WARNING' message is displayed: 'Include a goto configure mode button in your application if you need to access the configuration mode screens. Include a shutdown button in your application if you want to be able to shut it down.' A progress bar shows 53% completion. A 'Downloading...' dialog box is overlaid, showing the same source and destination paths and a progress bar.
- Bottom Screenshot:** The 'Download' tab is active. The 'Source file' is 'C:\Users\Public\Documents\RSView Enterprise\ME\Runtime\FTVME_APP.mer'. The 'Download as' field is 'FTVME_APP'. The 'Destination storage type' is 'Internal Storage'. A 'WARNING' message is displayed: 'Include a goto configure mode button in your application if you need to access the configuration mode screens. Include a shutdown button in your application if you want to be able to shut it down.' A 'Transfer Utility' dialog box is overlaid, showing 'The download has completed successfully.' and an 'OK' button.

10. After the download process, you will notice that your PanelView reboots, and loads the application we downloaded to it. Play around with the application running on your PanelView and make sure everything works without errors. If everything works, you have completed this lab. If you have any errors, please seek the help of one of the instructors.

Lab 11 Review:

1. What does .MER stand for?
2. What does .MED stand for?
3. What happens at runtime if you don't check the box "Run application at start up"?

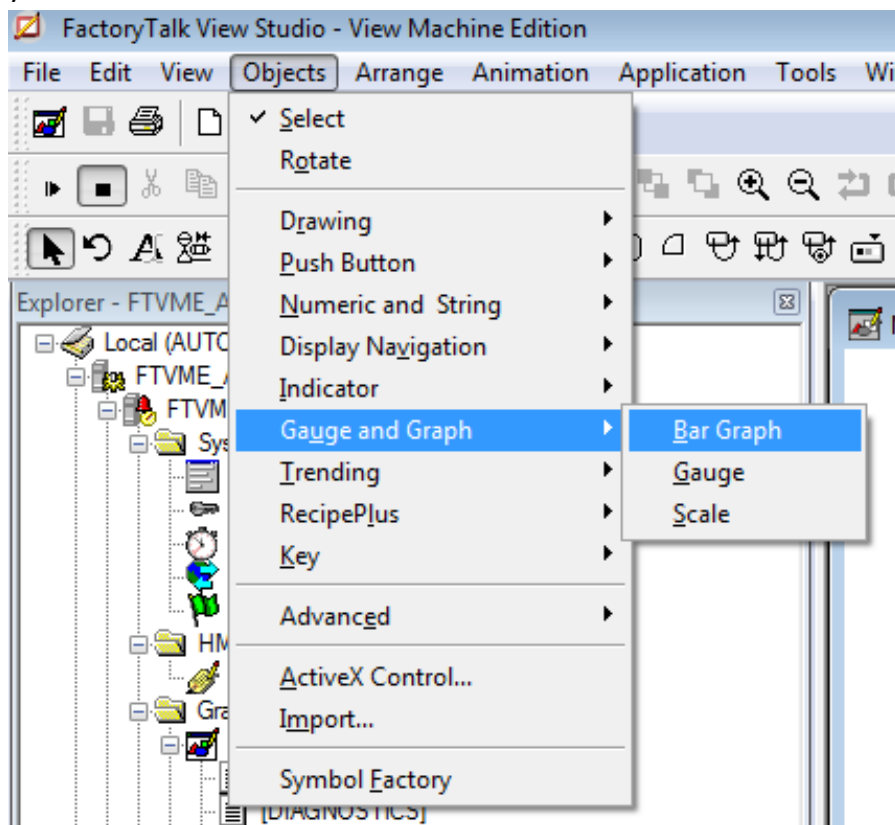


This now completes lab 11. Please wait for instructions to proceed further.

Lab 12: Creating a bar graph

Objective: To familiarize the user with the concept and implementation of bar graphs in a PanelView Plus application. This lab will walk the user through the complete process of creating and configuring a bar graph, as well as empower the user to apply this to his/her specific needs.

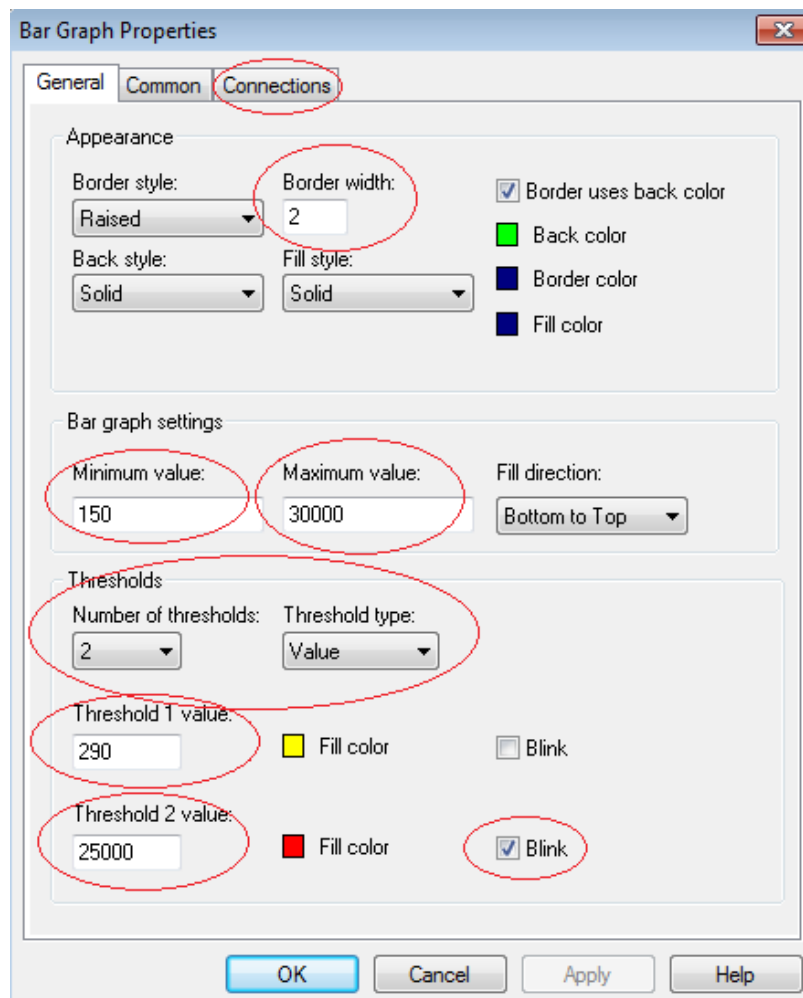
1. The first thing we need to do to create a bar graph on our display is to find it in the objects menu, draw one on our MAIN display and configure it. Start by going to “Objects→Gauge and Graph→Bar Graph”. Once you make this selection, draw a basic shape on the MAIN display and the properties window for the graph should open once you release the mouse button.



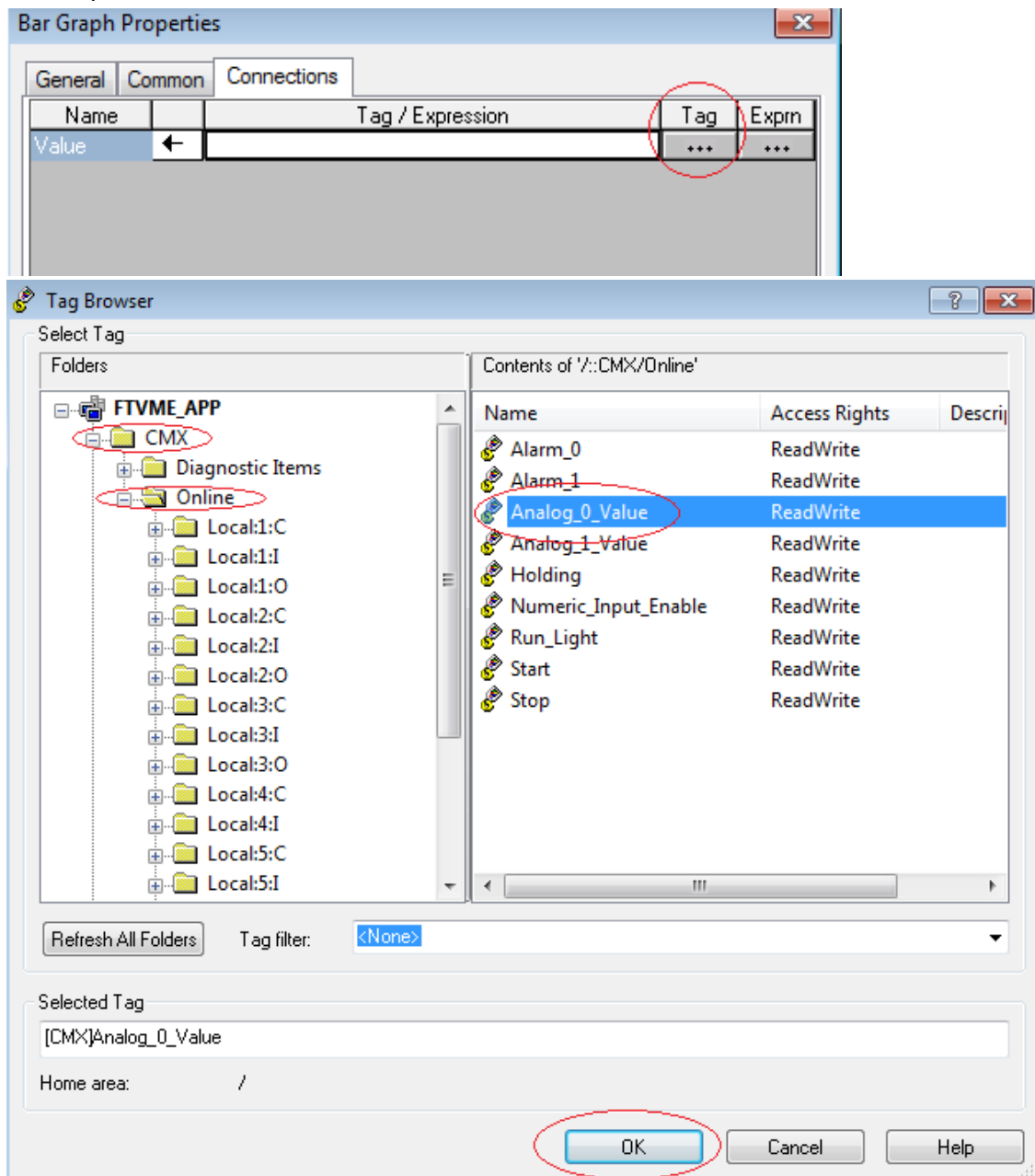
2. The default tab that opens up for the graph properties will be the “General” tab. From this tab we need to make a few changes. Enter the following information into the “General” tab of the bar graph properties window:

- Border Width: 2
- Maximum Value: 30000
- Minimum Value: 150
- Number of Thresholds: 2
- Threshold Type: Value
- Threshold 1 value: 290
- Threshold 2 value: 25000
- Check the “Blink” checkbox

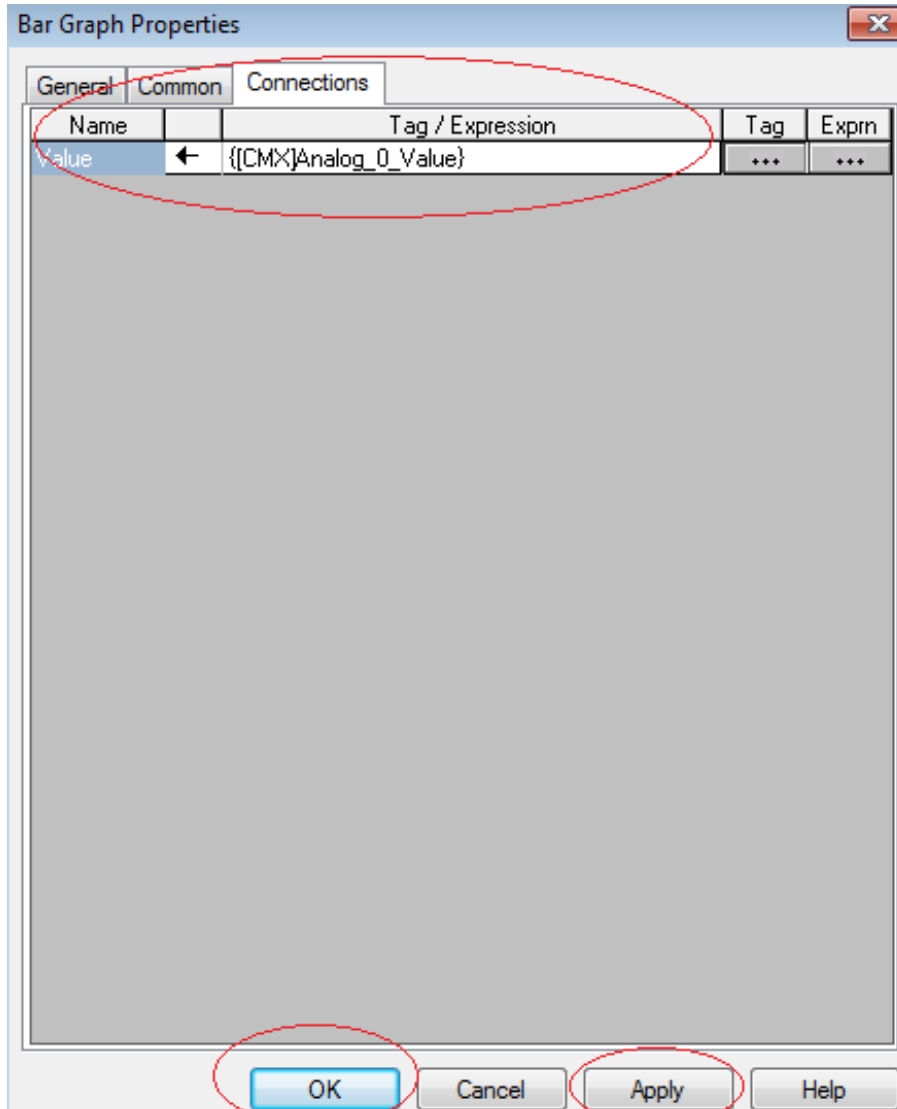
Your “General” tab within the bar graph properties window should now look like the image below. Once you have made these changes successfully, click over to the “Connections” tab.



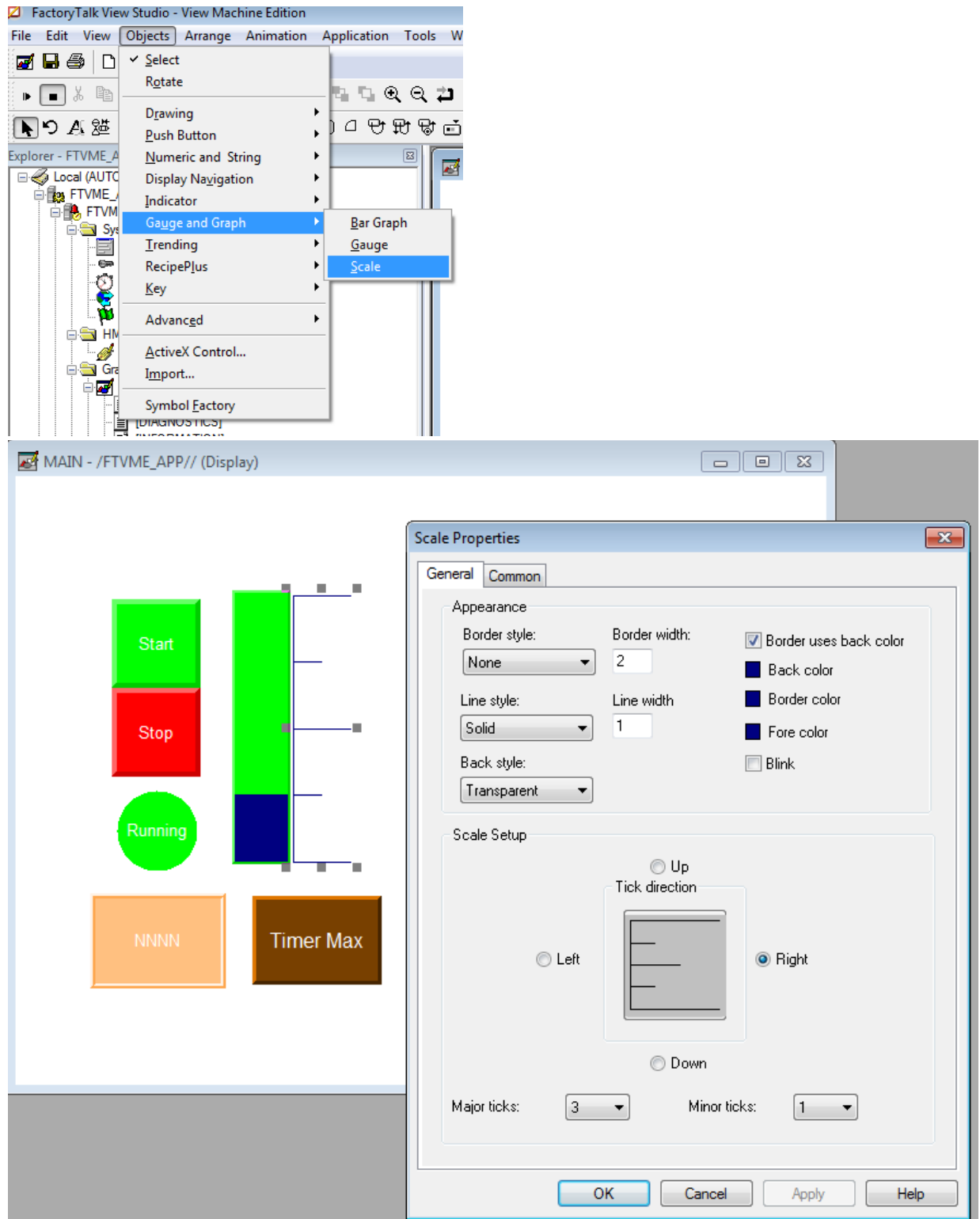
3. Once you are within the “Connections” tab of the bar graph properties window, select the tag browse button. Clicking the tag browse button will open the tag browser window, from which we will select our PLC tag. Within the Tag browser window, select the tag “[CMX]Analog_0_Value” which can be found in the location “CMX→Online→Analog_0_Value”. Once you have this tag highlighted, click the “OK” button at the bottom of the tag browser window. If you can’t see this tag, please seek the help of one of the instructors.



4. After clicking “OK” in step 3, you will have placed the tag “{[CMX]Analog_0_Value}” into the tag / expression field of the bar graph properties window. Your connections tab should look like the image below. Once you have the tag in place, click the “Apply” followed by the “OK” buttons at the bottom of the bar graph properties window.

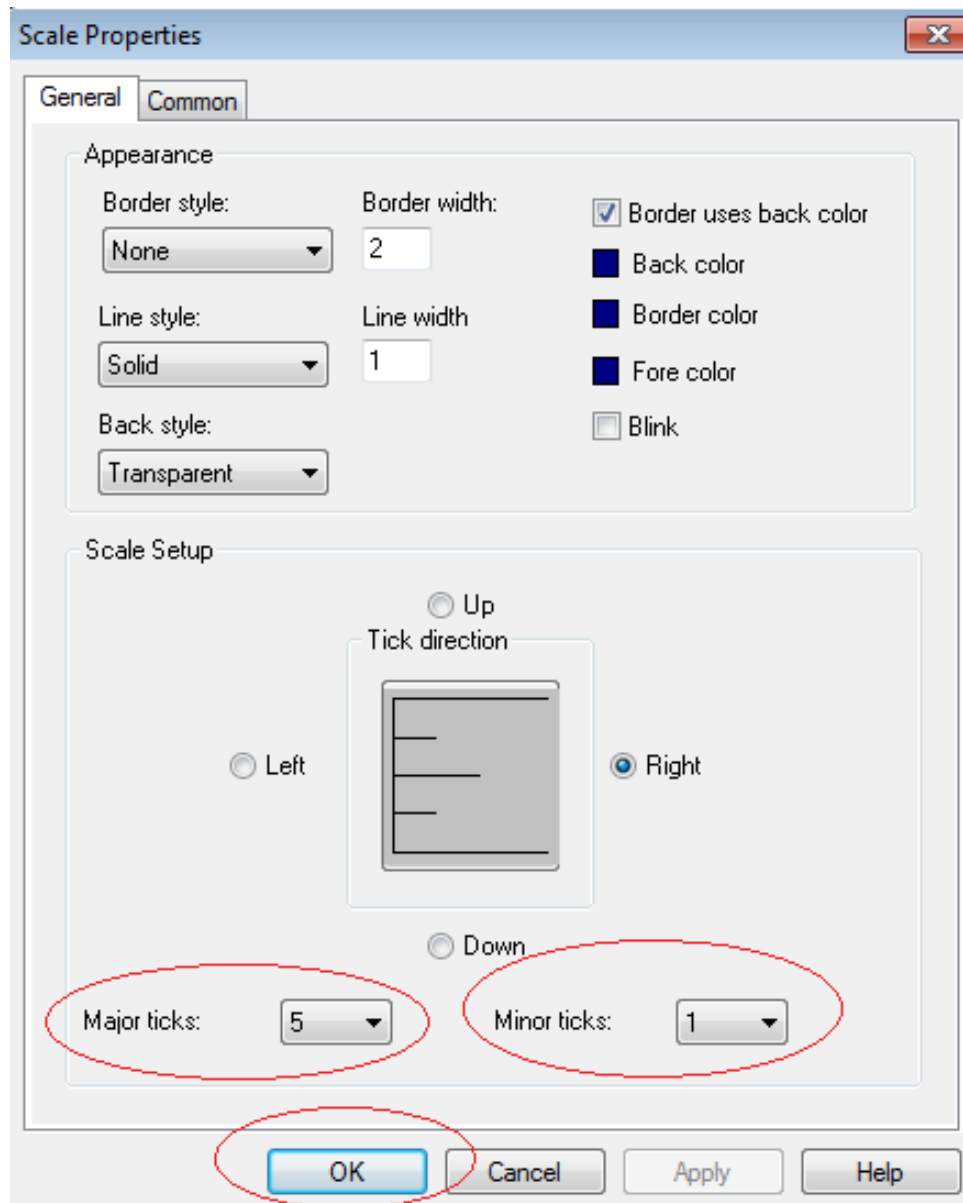


5. Now we are going to add scales to our bar graph so that we know what the values are as the bar moves up and down. You can find the scale by going to “Objects→Gauge and Graph→Scale”. Once you have selected the scale, draw a shape that matches your existing bar graph as shown below.

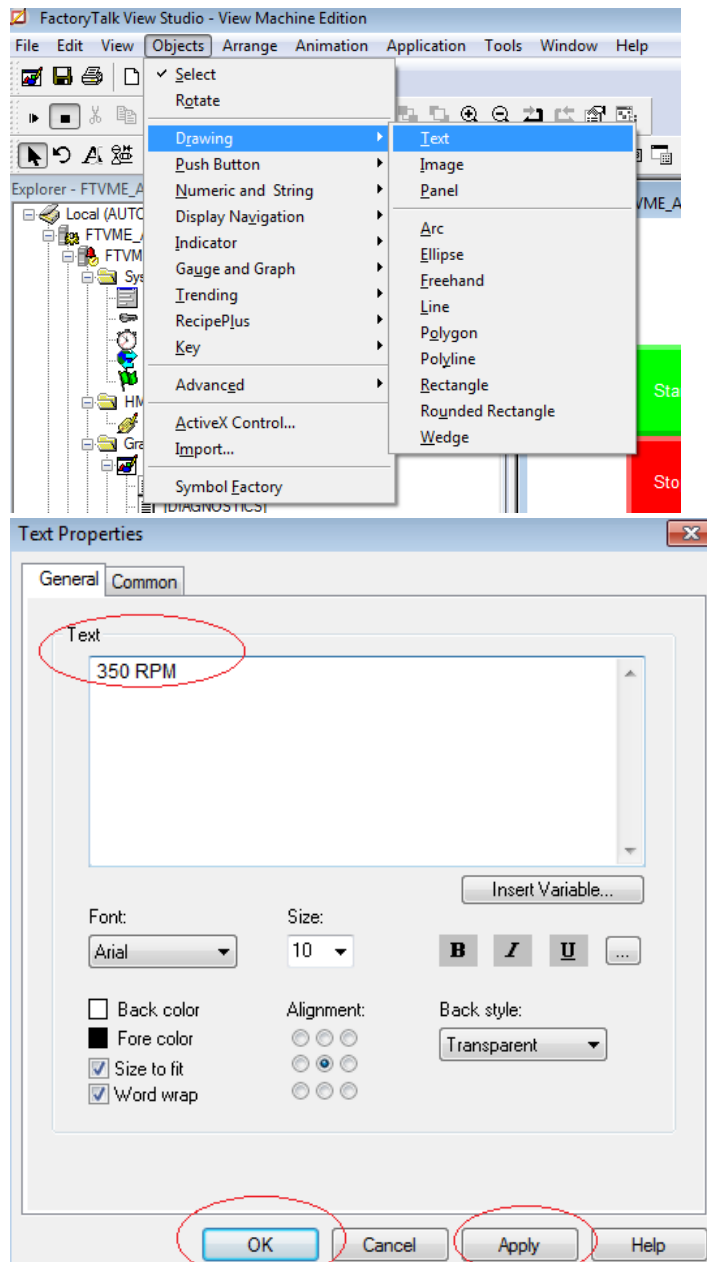


6. The default tab that will open for the scale properties window is the “General” tab. From this tab we need to make a couple changes. Enter in the following information into the “General” tab of the Scale Properties window.
- Major Ticks: 5
 - Minor Ticks: 1

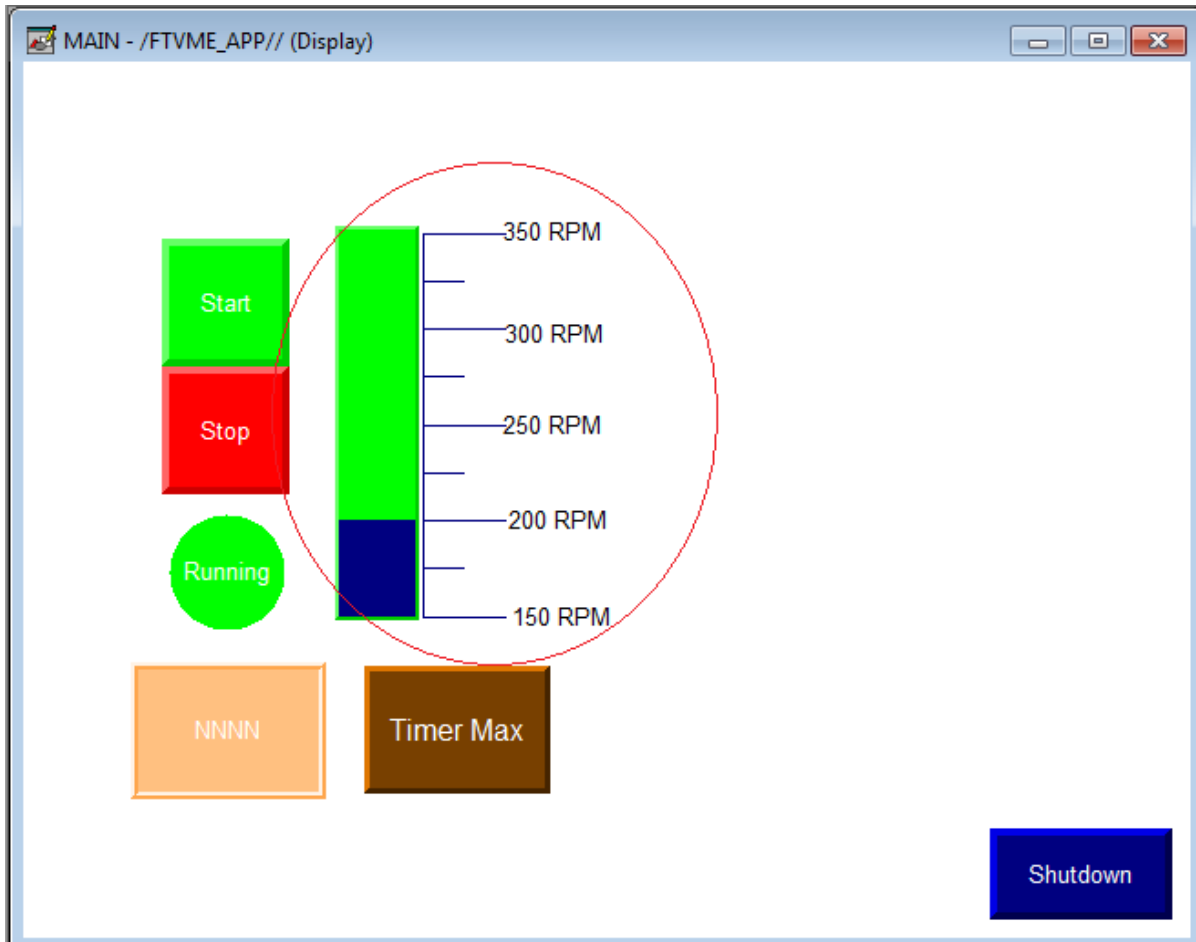
Your scale properties “General” tab should now look like the image below. Once you have successfully made these changes, click the “OK” button at the bottom of the properties window.



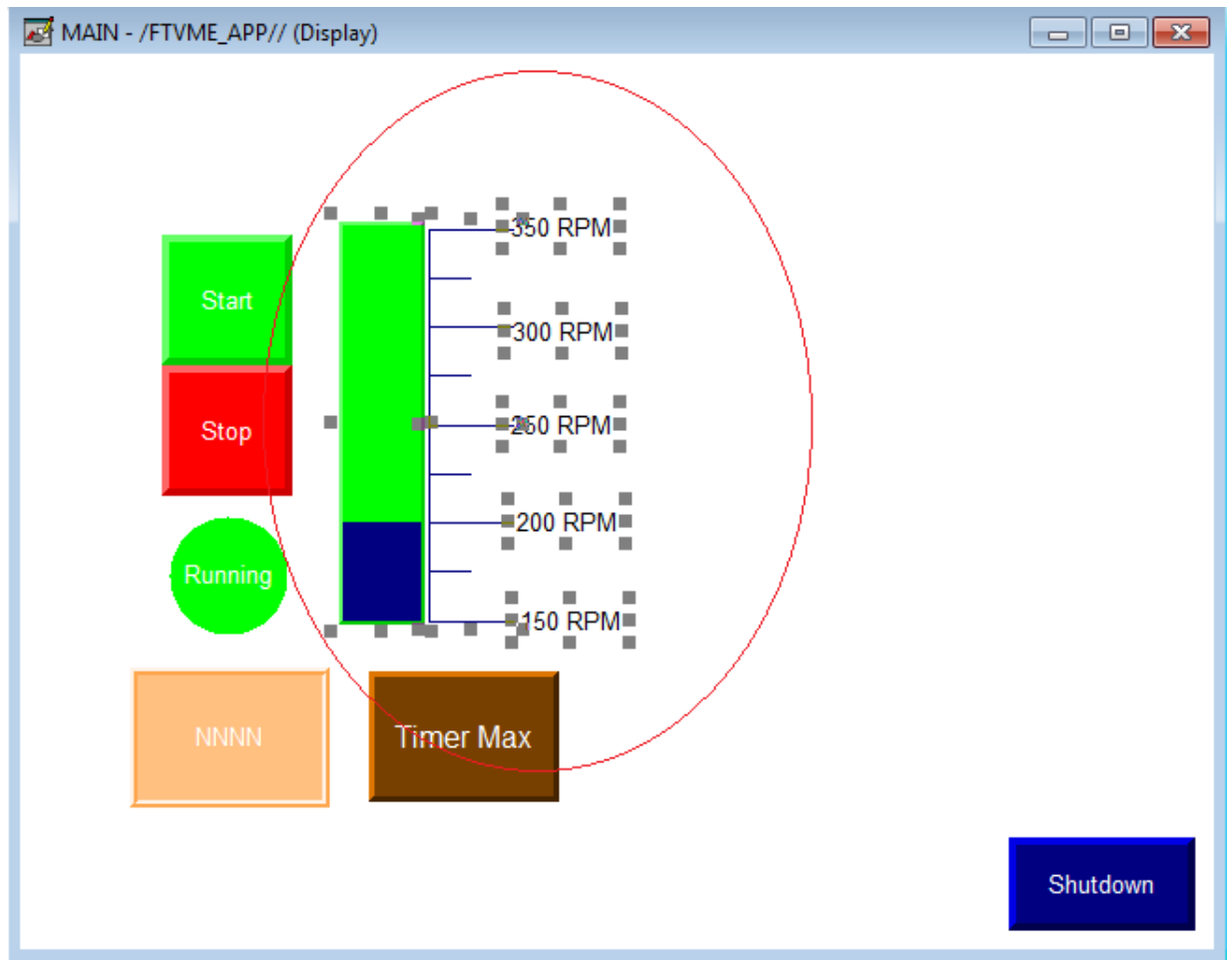
7. Now that we have configured our scale, it is time to add some values to our MAIN display so that the scale and bar graph make sense. We are going to add text objects next to each “Tick” on our scale so that we can understand what the bar graph is reading. Start by going to “Objects→Drawings→Text”, and draw a text box next to the top “tick” of our scale. In the text box, type “350 RPM” and then press the “Apply” and “OK” buttons at the bottom of the text properties window.



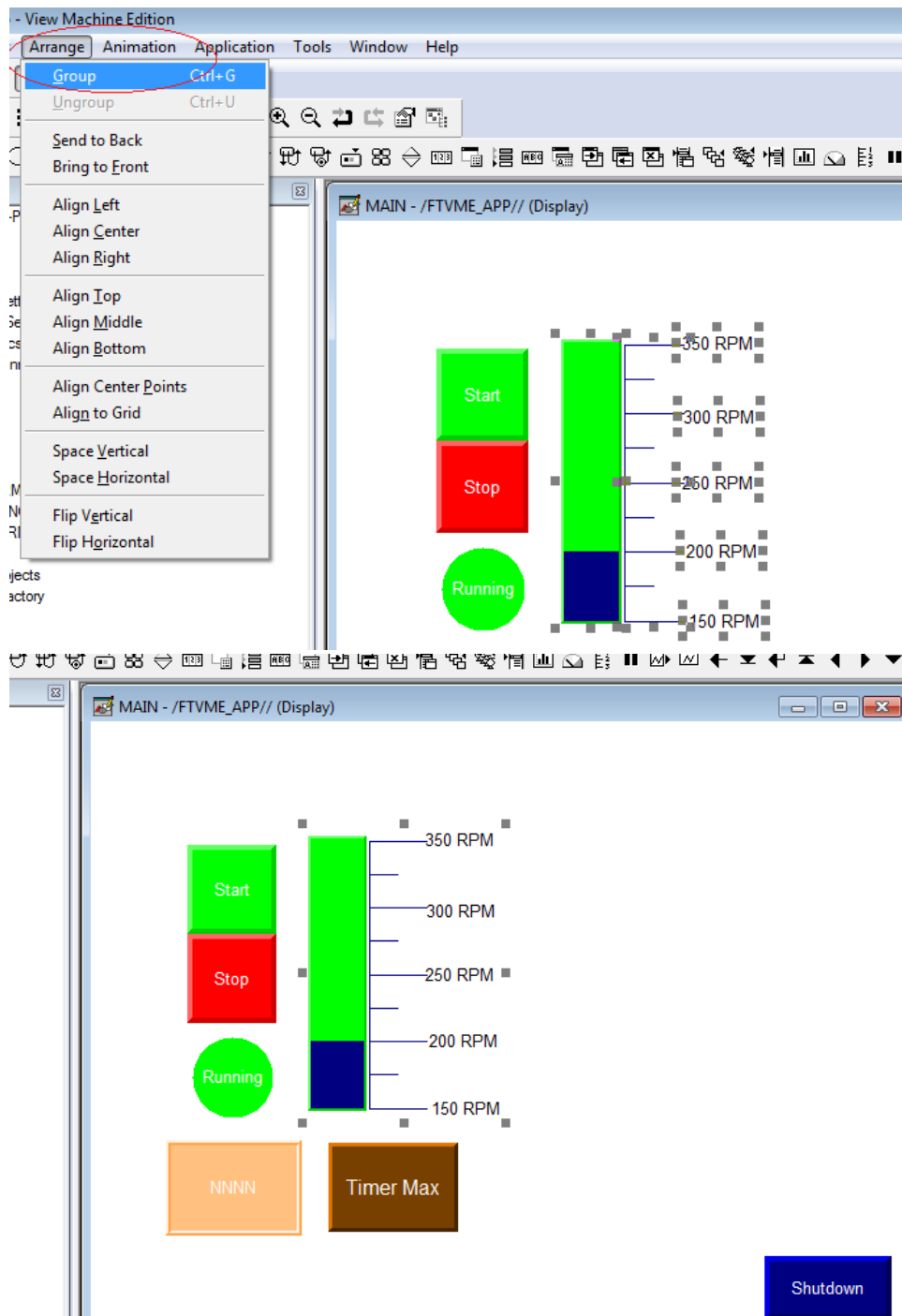
8. Repeat step 7, for each of the major “Ticks” on the scale, until your MAIN display looks like the image below.



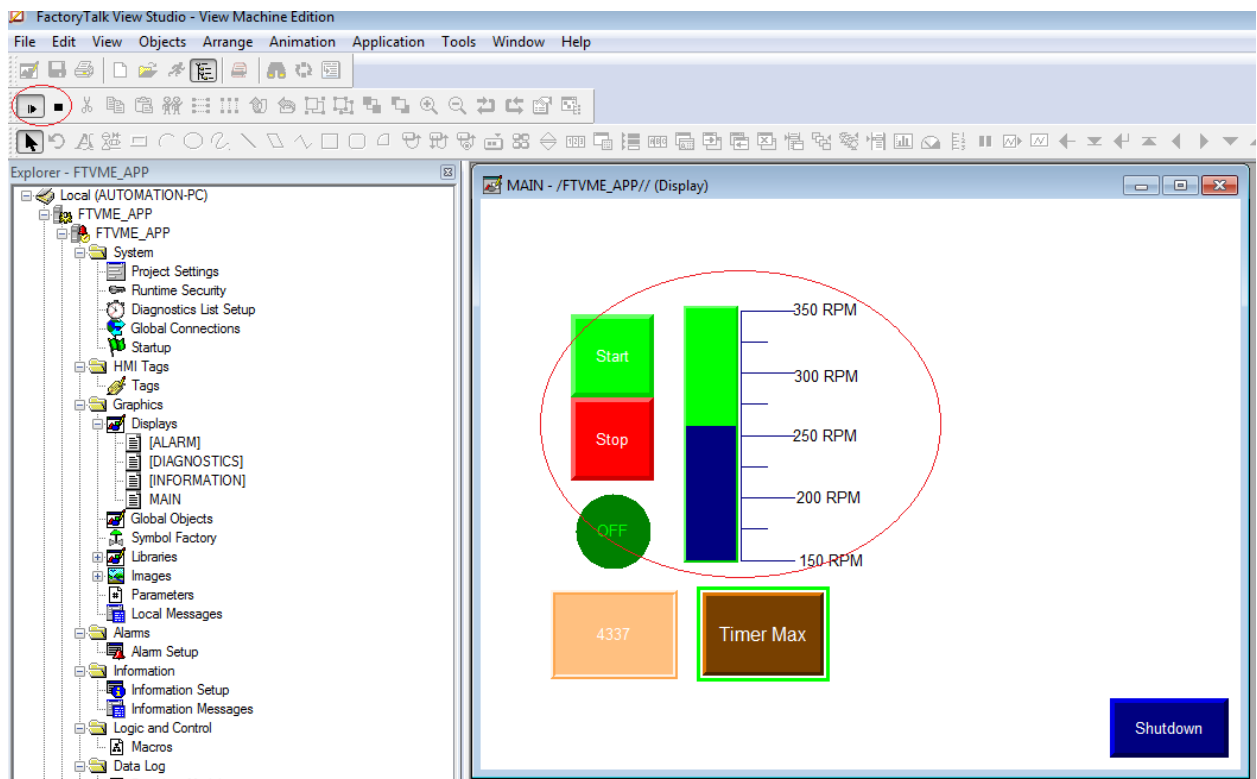
9. The next thing we will do is group objects together into a single object on the display. We will do this by clicking into open space to the top-left of our bar graph, then dragging the cursor across the bottom right of the text “150 RPM” so that our bar graph, scale, and text boxes are all selected at the same time as shown below.



10. With all the objects from step 9 highlighted, select “Arrange→Group”. Doing this will group all the objects that you have highlighted, together. This can also be done by pressing the ALT+G key on your keyboard.



11. Once you have the objects from step 10 grouped together, test your display by clicking the test display button at the top left corner of the toolbar. Make sure when you rotate potentiometer AI0 in your demo case, that the bar graph indicates a change. If you see your bar graph move up and down with rotation of your potentiometer, your lab is complete. If you do not see any movement within your bar graph, please seek the help of an instructor. Click the stop button located directly to the right of the test display button.



Lab 12 Review:

1. Does the scale have a min and max value that you can enter in the properties window?
If so, which tab?
2. How do you add numbers to your scale?
3. What is the max number of thresholds available on a bar graph?

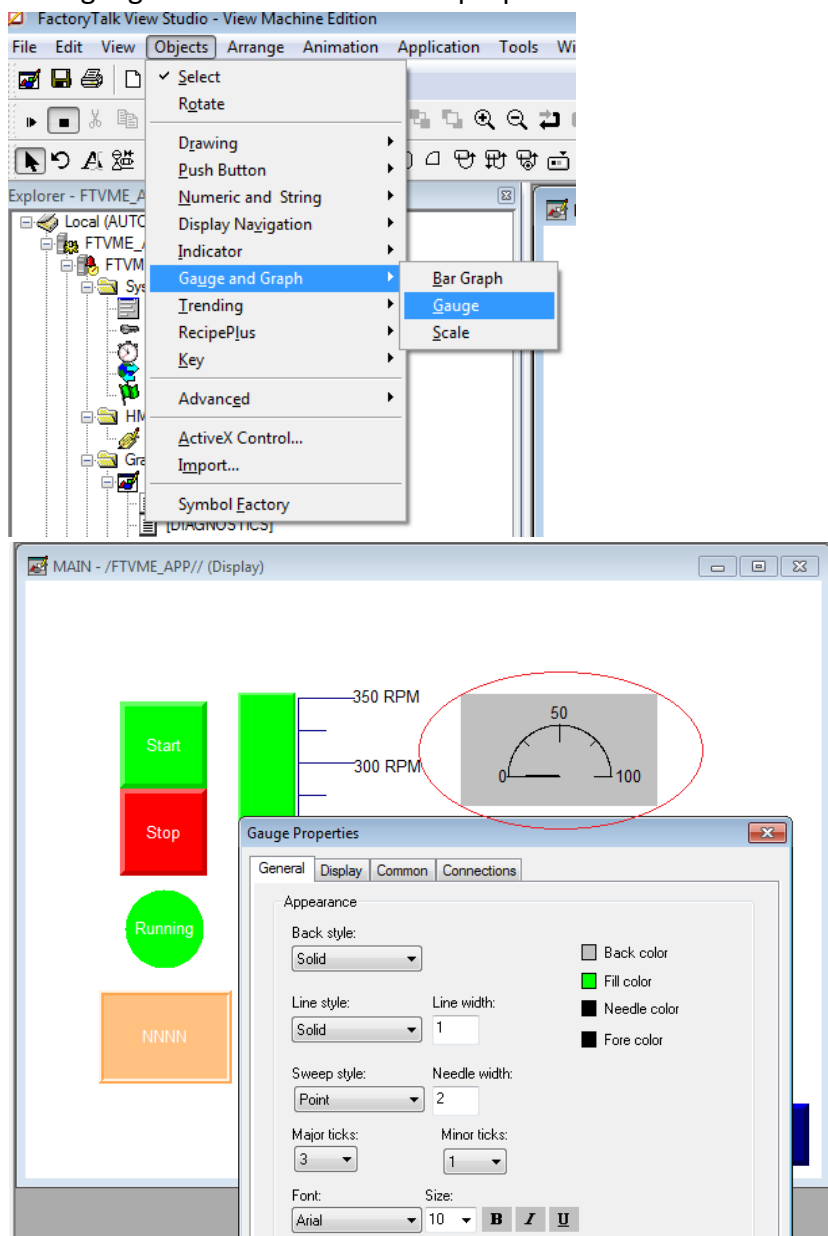


This completes lab 12. Please wait for instructions to proceed further.

Lab 13: Creating a Gauge

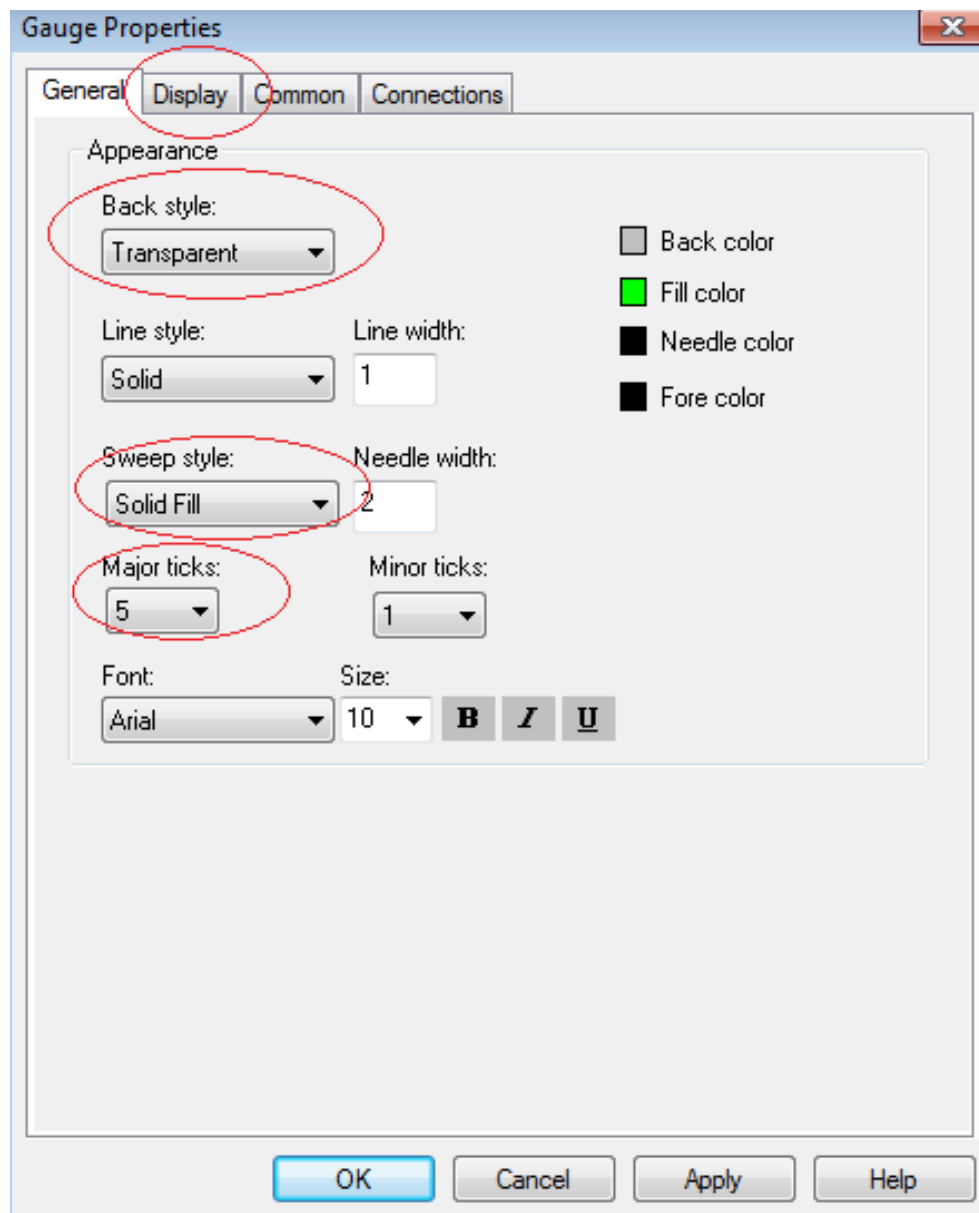
Objective: to introduce the concept of a gauge object on the PanelView Plus display. This includes creation, configuration, and implementation. At the end of this lab, the user will be able to create a gauge that displays valid process information, which could be used in many different applications.

1. The first thing we need to do in order to add a gauge to our MAIN display is to find it within the objects list and draw one on our MAIN display. You can find the gauge object from “Objects→Gauge and Graph→Gauge”. Once you select the gauge object, draw a basic gauge on the screen and the properties window will automatically open.



2. The default tab that is opened within the gauge properties window is the “General” tab. Within this tab we need to make a few changes. Enter the following information into the gauge properties “General” tab.
- Back Style: Transparent
 - Sweep Style: solid fill
 - Major Ticks: 5

Once you have successfully entered this information into the “General” tab of the gauge properties window, your “General” tab should look like the image below. Once your “General” tab has been completed, click over to the “Display” tab.



3. From the “Display” tab there are a few changes we need to make. Enter in the following information into the “Display” tab of the gauge properties window.

- Minimum Value: 0
- Maximum Value: 30000
- Threshold Type: Percentage
- Threshold 1 Value: 50
- Threshold 2 Value: 75
- Number of Thresholds: 2

Once you have successfully made these changes, your “Display” tab should look like the image below. If you have made these changes, click over to the “Connections” tab.

The screenshot shows the 'Gauge Properties' dialog box with the 'Display' tab selected. The 'Connections' tab is also visible and circled in red. The 'Value settings' section has 'Minimum value' set to 0 and 'Maximum value' set to 30000, both circled in red. The 'Legend settings' section has 'Show legend' checked, 'Digits after decimal' set to 0, and a black 'Legend color' swatch. The 'Threshold settings' section has 'Number of thresholds' set to 2, 'Threshold type' set to 'Percentage', 'Threshold 1 value' set to 50, and 'Threshold 2 value' set to 75, all circled in red. There are also color swatches for 'Fill color' (yellow and red) and 'Blink' checkboxes for each threshold.

Gauge Properties

General Display Common **Connections**

Value settings

Minimum value: 0 Maximum value: 30000

Legend settings

☒ Show legend Digits after decimal: 0 Legend color

Threshold settings

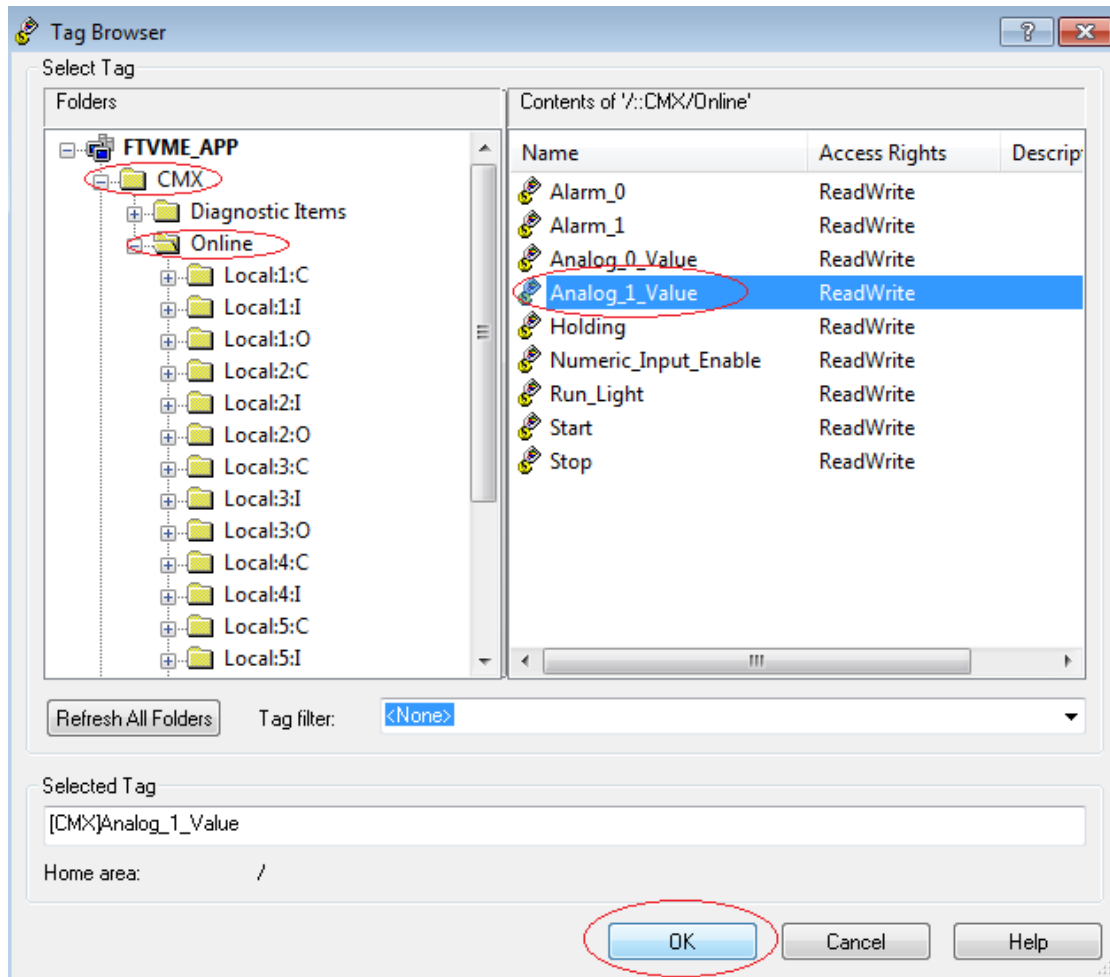
Number of thresholds: 2 Threshold type: Percentage

Threshold 1 value: 50 Fill color Blink

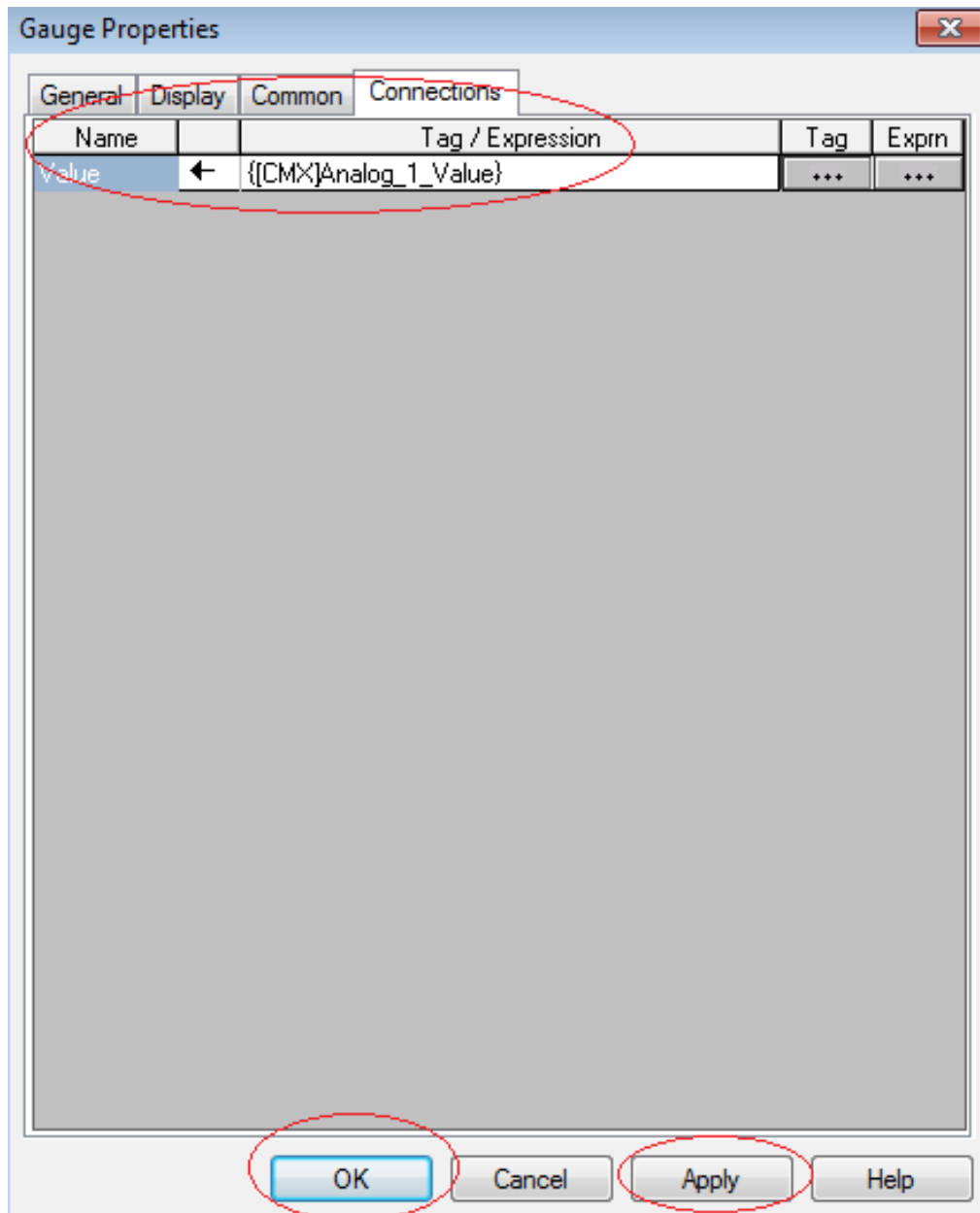
Threshold 2 value: 75 Fill color Blink

OK Cancel Apply Help

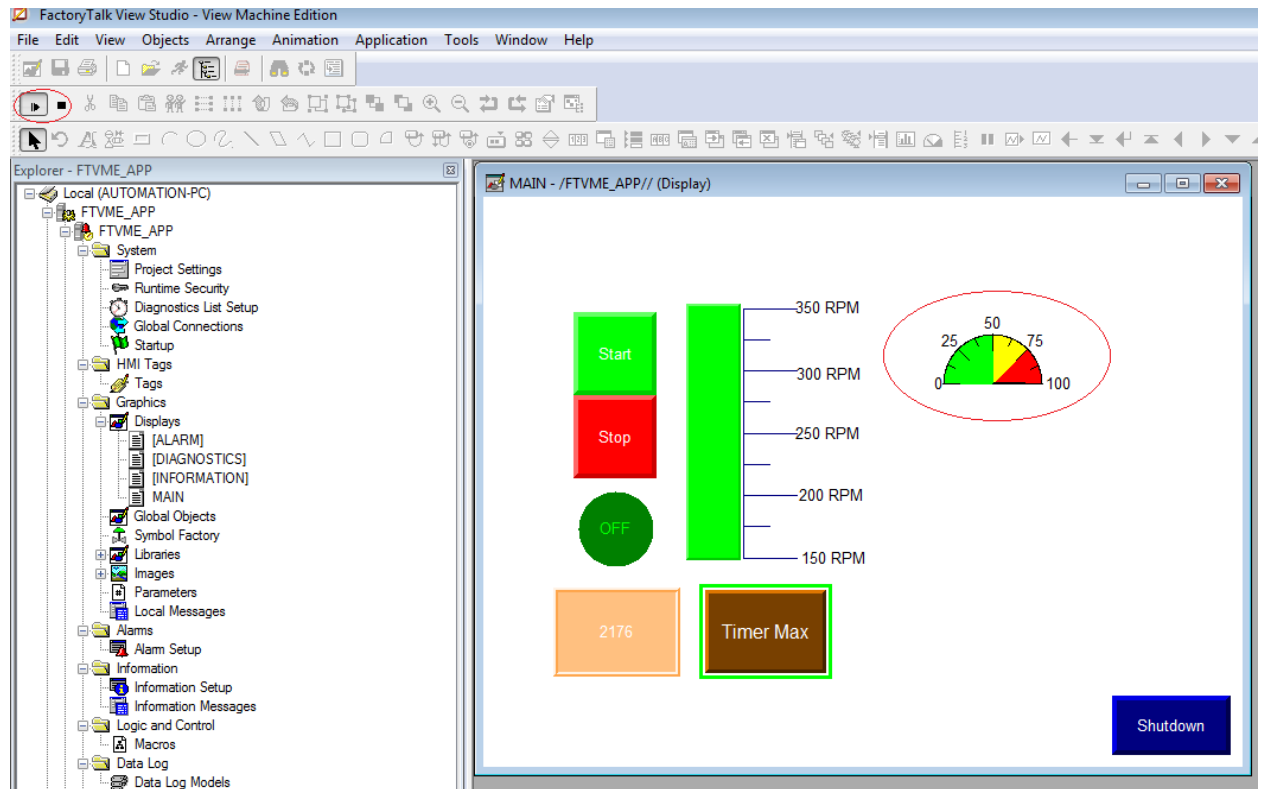
4. Within the “Connections” tab of the gauge properties window, select the tag browse button which will open the tag browser. From the tag browser window, highlight tag {[CMX]Analog_1_Value}, which can be found at “CMX→Online→Analog_1_Value”. Once you have this tag highlighted, click the “OK” button at the bottom of the tag browser window which will assign that tag to the tag / expression field of our gauge properties window.



5. After clicking “OK” from step 4, you should now see the tag “`{{[CMX]Analog_1_Value}}`” located within the tag / expression field of the gauge properties window. If you do not see this tag, please repeat step 4, or seek the help of one of the instructors. Once the correct tag is in the tag / expression field, click the “Apply” then the “OK” buttons at the bottom of the gauge properties window.



6. With our newly created gauge on the MAIN display, it's time for testing. Click the test display button located in the upper left-hand corner of the toolbar and make sure that as you turn the potentiometer labeled "AI1" the gauge goes up and down respectively. If your gauge operates as it should, please click the stop button located directly to the right of the test display button and your lab is complete. If your gauge does not operate with the turning of the potentiometer, please seek the help of one of the instructors.



Lab 13 Review:

1. What is the maximum number of thresholds available for the gauge?
2. What are the 2 different sweep styles available for the gauge?
3. In the gauge property window, what tab can you set the minimum and maximum values at?

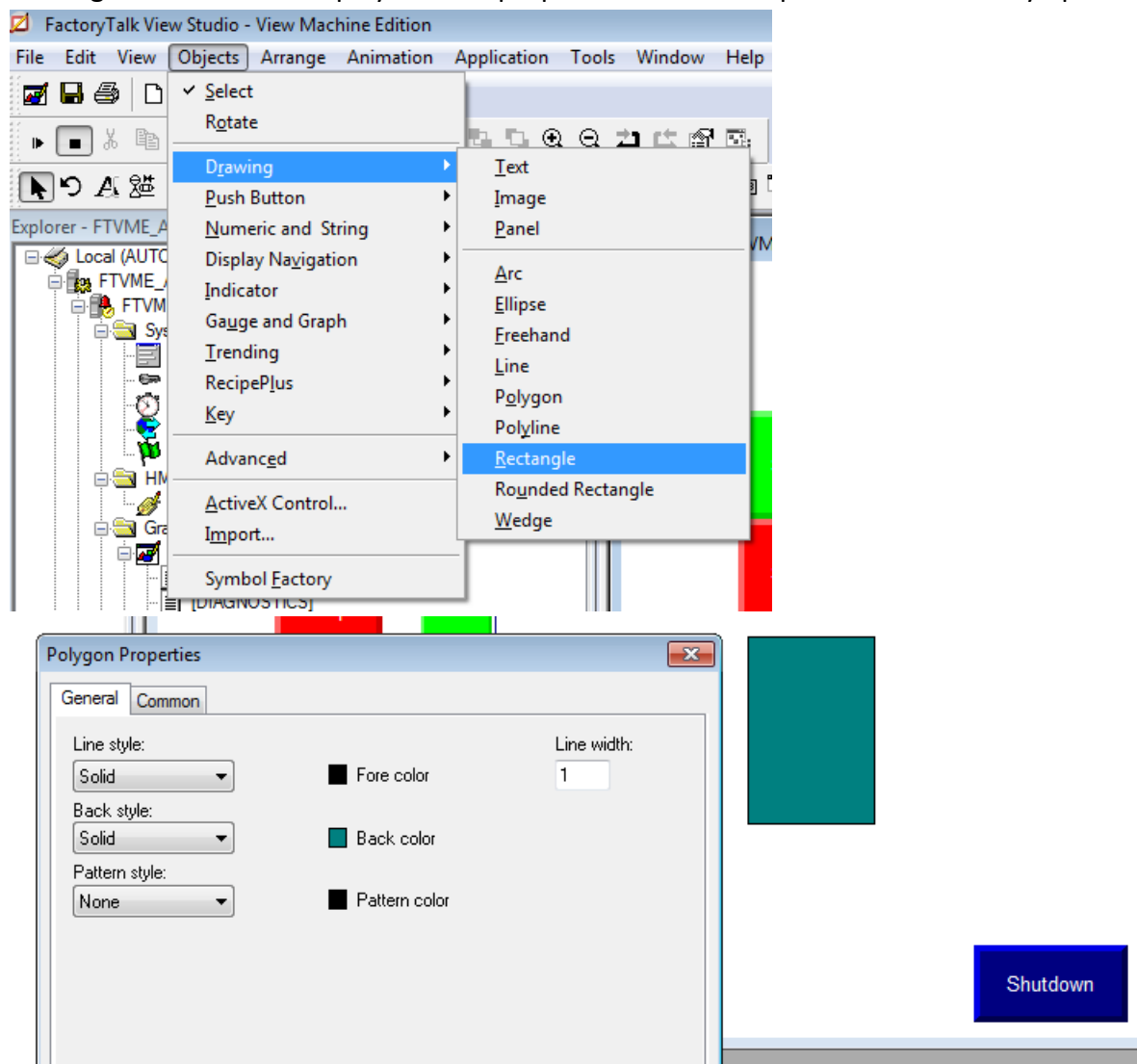


This completes lab 13. Please wait for instructions to proceed further.

Lab 14: Working with basic shapes

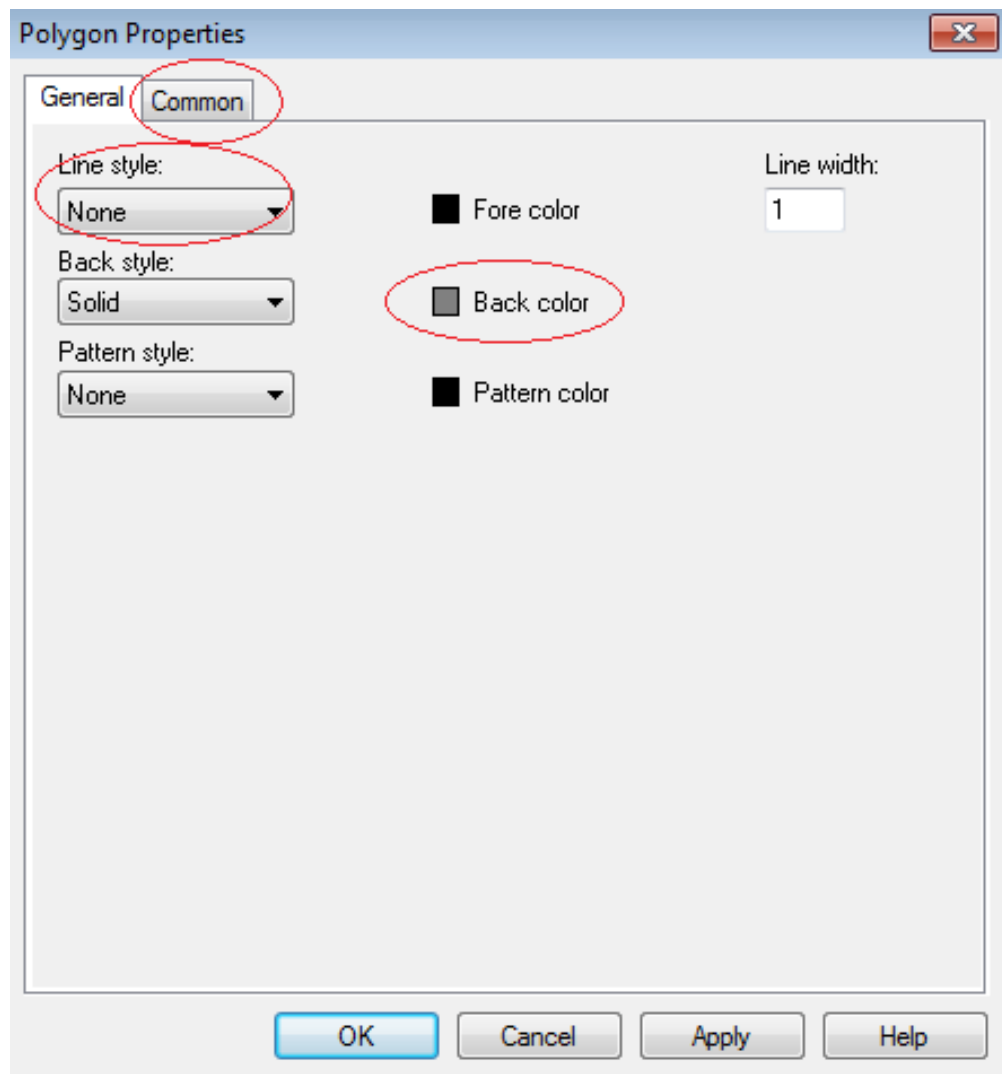
Objective: To provide a general overview of creating and configuring basic shapes on the PanelView Plus display. By the end of this exercise, the user should be able to manipulate these objects in an efficient and useful manner.

1. The first thing we will create using basic shapes, is a rectangle. You can find this shape by going to “Objects→Drawing→Rectangle”. Once you have this selected, draw a basic rectangle on the MAIN display and the properties for that shape will automatically open.



2. The first tab that will be open, by default, within the Polygon properties window is the “General” tab. Within this tab, there are a couple changes we need to make. Enter the following information into the “General” tab of the properties window:
- Line Style: None
 - Back Color: Gray

Your “General” tab should now look like the image below. Once you have successfully made these changes, click over to the “Common” tab.



3. From the “Common” tab, there are a couple changes we need to make. Enter in the following information within the “Common” tab of the polygon properties window:
- Height: 180
 - Width: 110

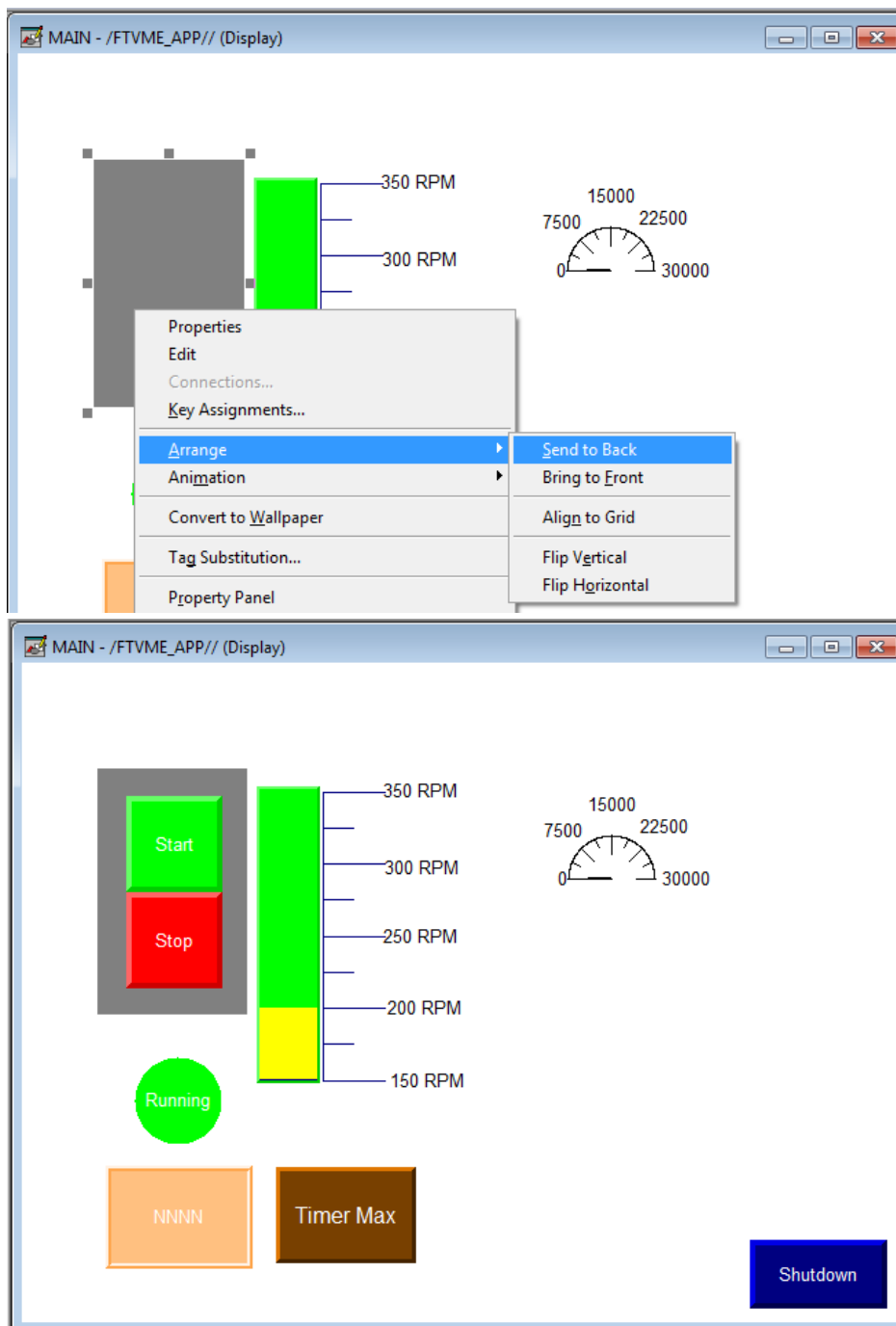
Once you have made the changes successfully, click the “Apply” followed by the “OK” buttons at the bottom of the properties window.

The image shows a screenshot of the "Polygon Properties" dialog box, specifically the "Common" tab. The dialog box has a title bar with a close button (X). Below the title bar are two tabs: "General" and "Common", with "Common" being the active tab. The "Common" tab contains several input fields and a checkbox. The "Size" section has "Height" and "Width" fields, both of which are circled in red. The "Height" field contains the value "180" and the "Width" field contains the value "110". The "Position" section has "Top" and "Left" fields, with values "233" and "358" respectively. Below these is an "Other" section containing a "Name" field with the value "Polygon1" and a checked checkbox labeled "Visible". At the bottom of the dialog box are four buttons: "OK", "Cancel", "Apply", and "Help". The "OK" and "Apply" buttons are circled in red.

Section	Property	Value
Size	Height	180
	Width	110
Position	Top	233
	Left	358
Other	Name	Polygon1
	Visible	☒

Buttons: OK, Cancel, Apply, Help

- The next step is to position the newly created rectangle over the top of the start / stop push buttons. The goal is to make this look like an enclosure for the buttons. Once you have the rectangle positioned over the start / stop buttons, right-click on the rectangle and select “Arrange→Send to back”. This will send the rectangle behind the start/stop buttons.



Lab 14 Review:

1. In reference to position on the display where is point 0,0?
2. When you right click on an object, what are the five different options when you select arrange?
3. Can you resize an object in the properties window? Is there another less precise way to resize?

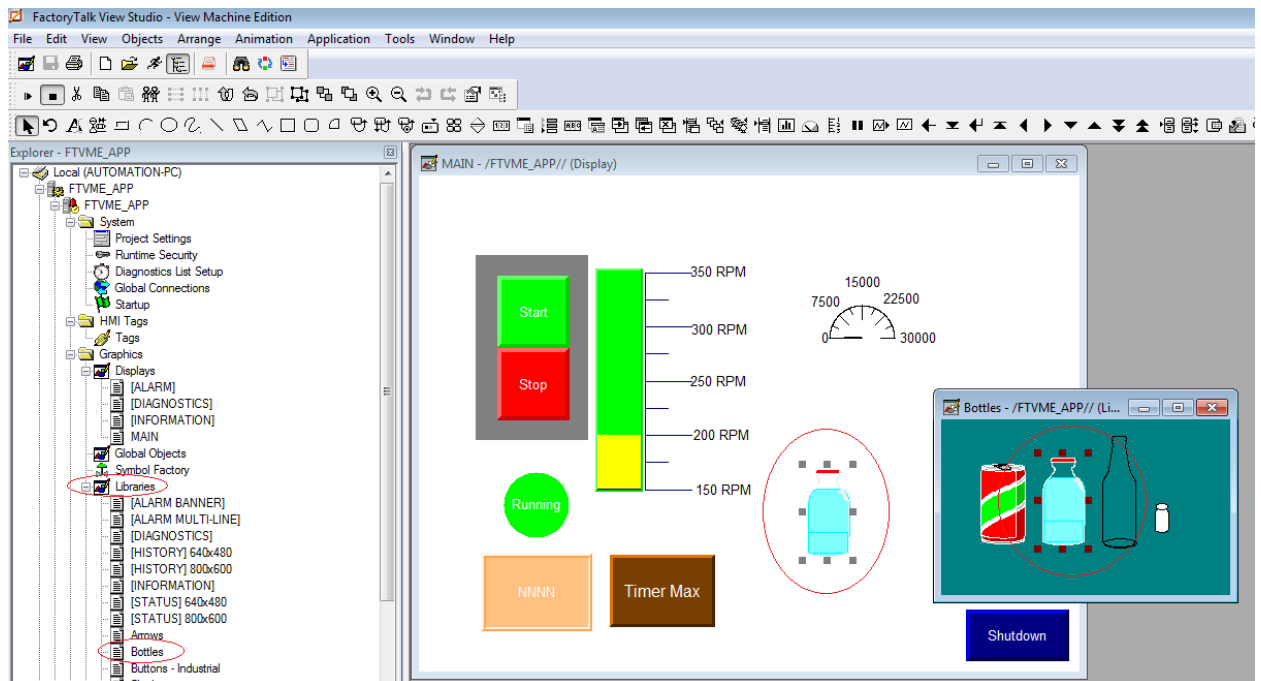


This completes lab 14. Please wait for instructions to proceed further.

Lab 15a: Working with Graphics

Objective: To provide a general overview of manipulating and configuring graphics on the PanelView Plus display. This includes using a graphic to represent a pushbutton, as well as other concepts.

1. The first thing we are going to do, in order to use one of the graphics out of the library, is click the “+” next to libraries in the explorer window. Double click on bottles and then drag and drop one of them from the window onto your MAIN display.

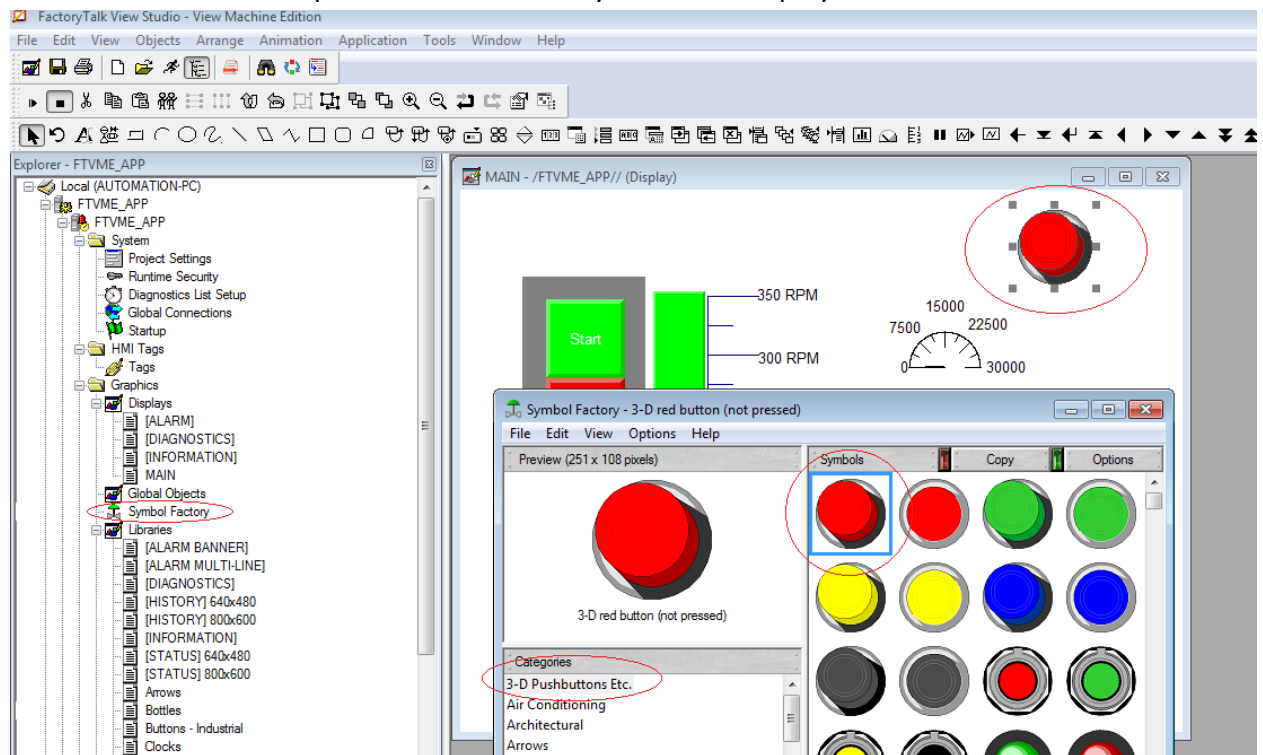


2. Close out of the bottles pop-up. This completes lab 15a.

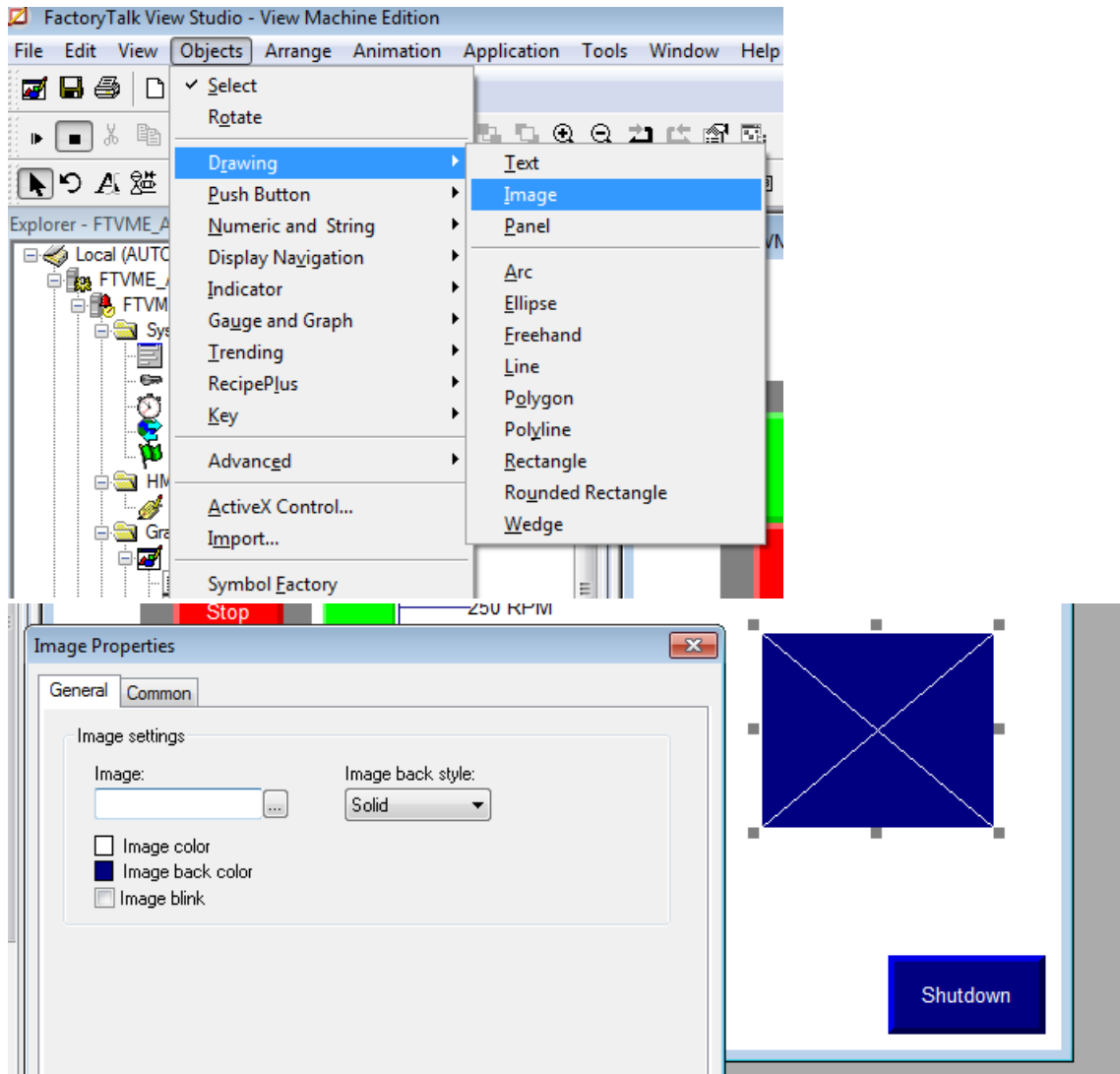
Lab15b: Using Symbol Factory

Symbol factory is only available in FactoryTalk View 6.0 and newer. Add an object with the drag and drop method and add an object with the image properties

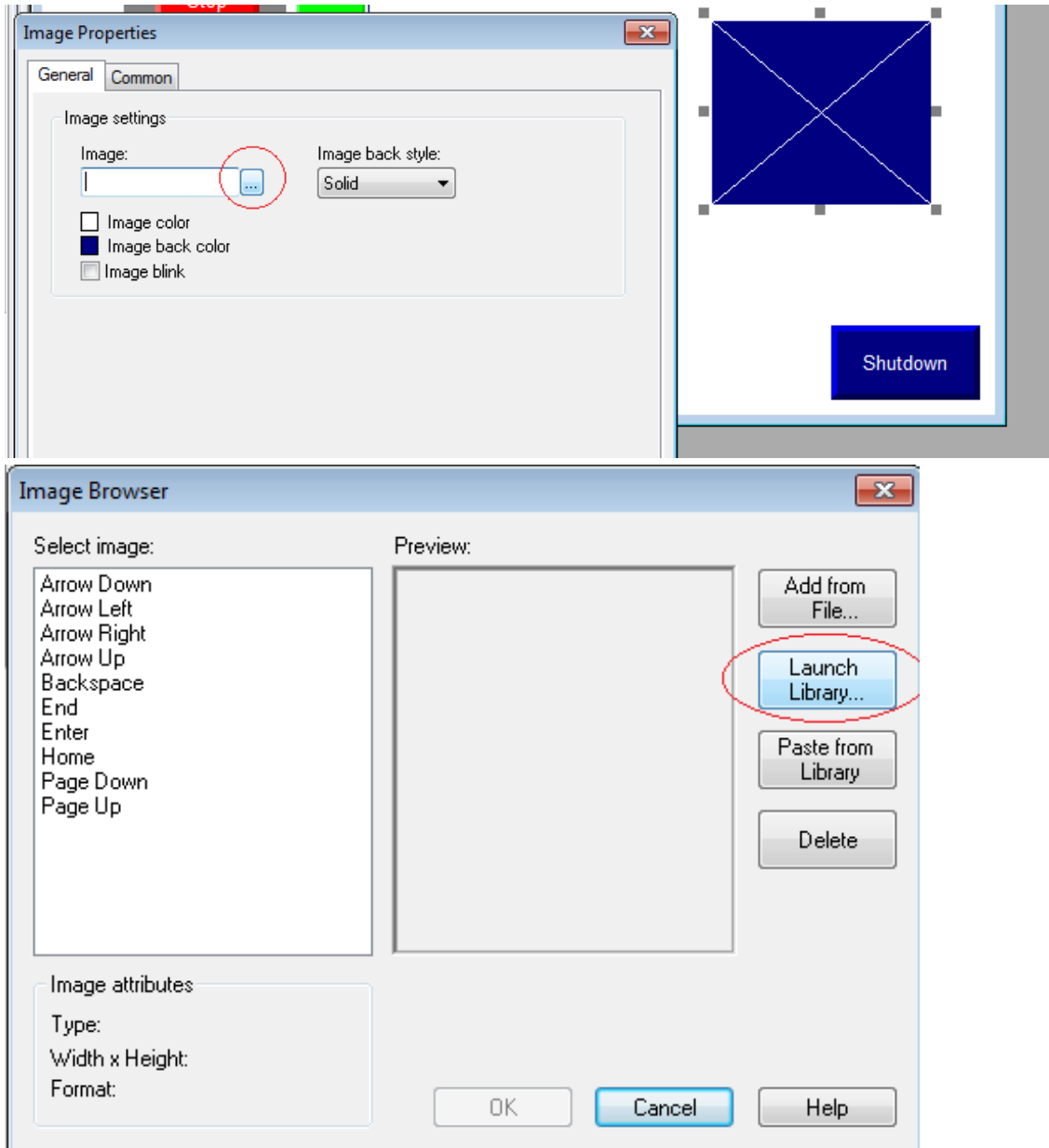
1. To utilize the objects in symbol factory, start by double clicking on the “Symbol Factory” icon located in the project explorer tree. The first category that will open up with the symbol factory is “3-D pushbuttons Etc”. Drag and drop the push button from the upper left-hand corner of the push button list onto your MAIN display.



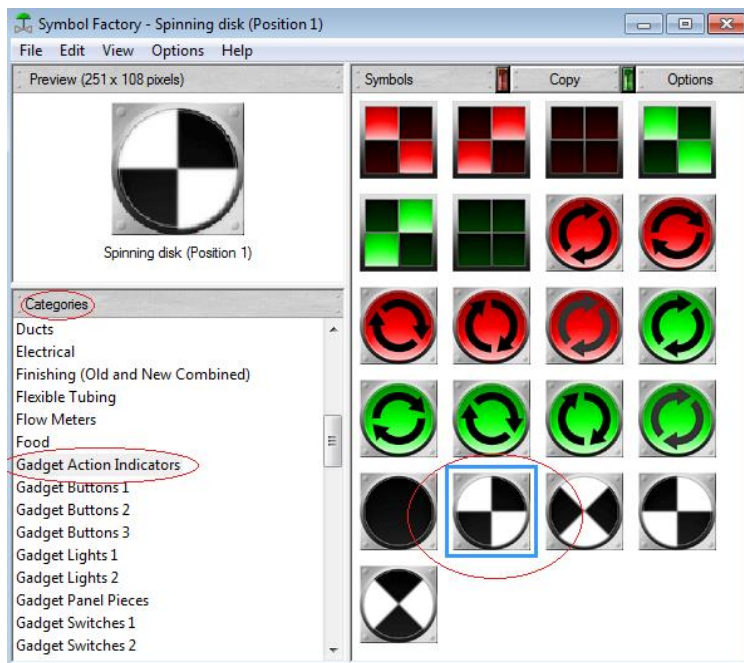
- The next object we will explore is an image. Start by closing out of the symbol factory, then go to “Objects→Drawing→Image”. Draw a basic shape on your MAIN display in an open space and the properties window will open. If the properties window doesn’t open, double click the image shape you just drew.

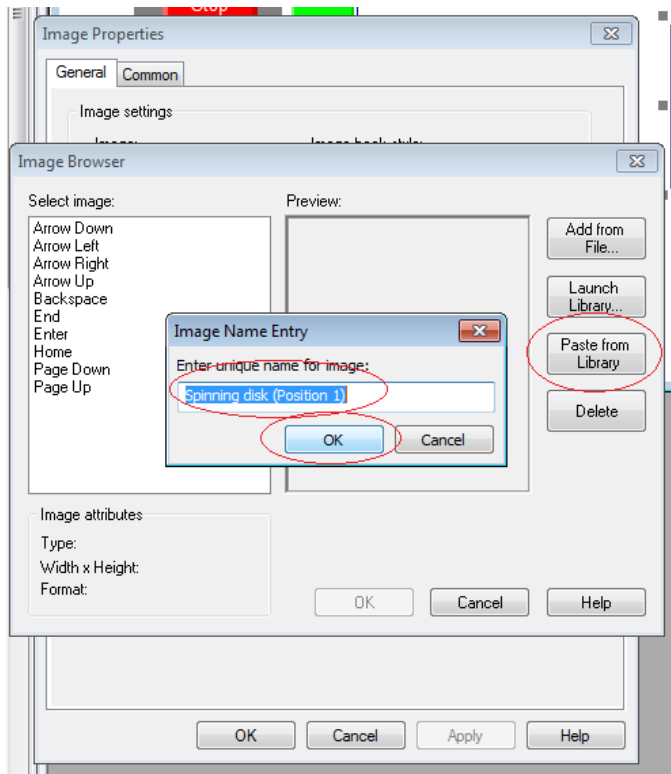


3. The first tab open by default within the image properties window, is the “General” tab. From this tab we will browse for an image by clicking the “browse” button, followed by the “Launch Library” button within the subsequent window as shown in the images below.

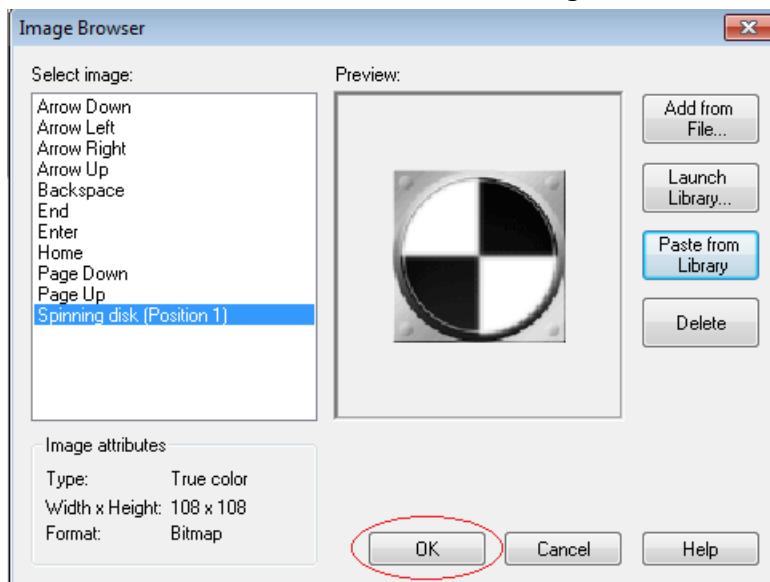


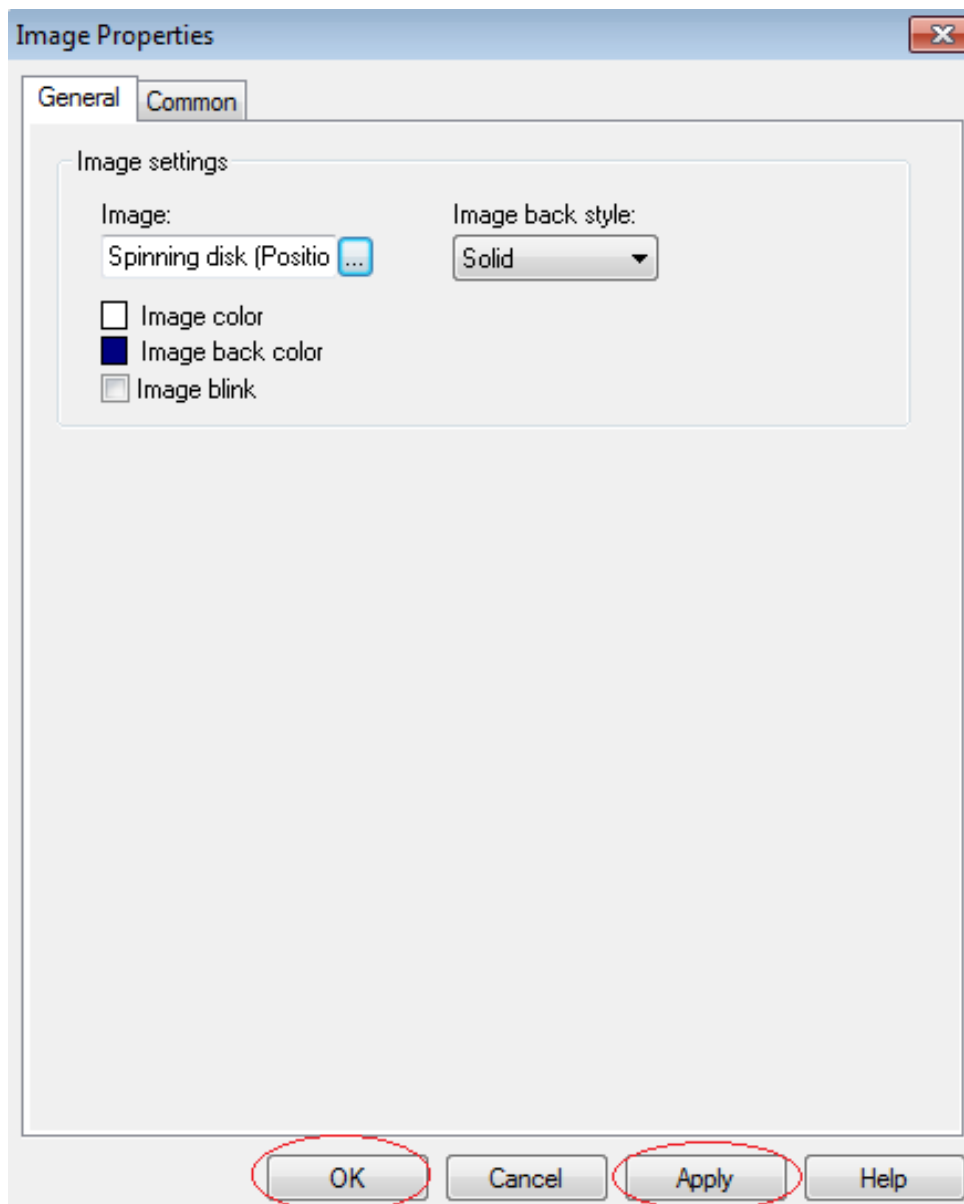
4. Once the library loads, scroll down to the “Gadget Action Indicators” category from the categories list. Once you have this highlighted, select a symbol, then double-click on it so that you are brought back to the image browser window. Select the “paste from library” button and you will see the symbol you chose in the preview window as shown below. Leave the image name default.





5. From the image browser window, select the “OK” button followed by the “Apply” and “OK” buttons at the bottom of the following window.

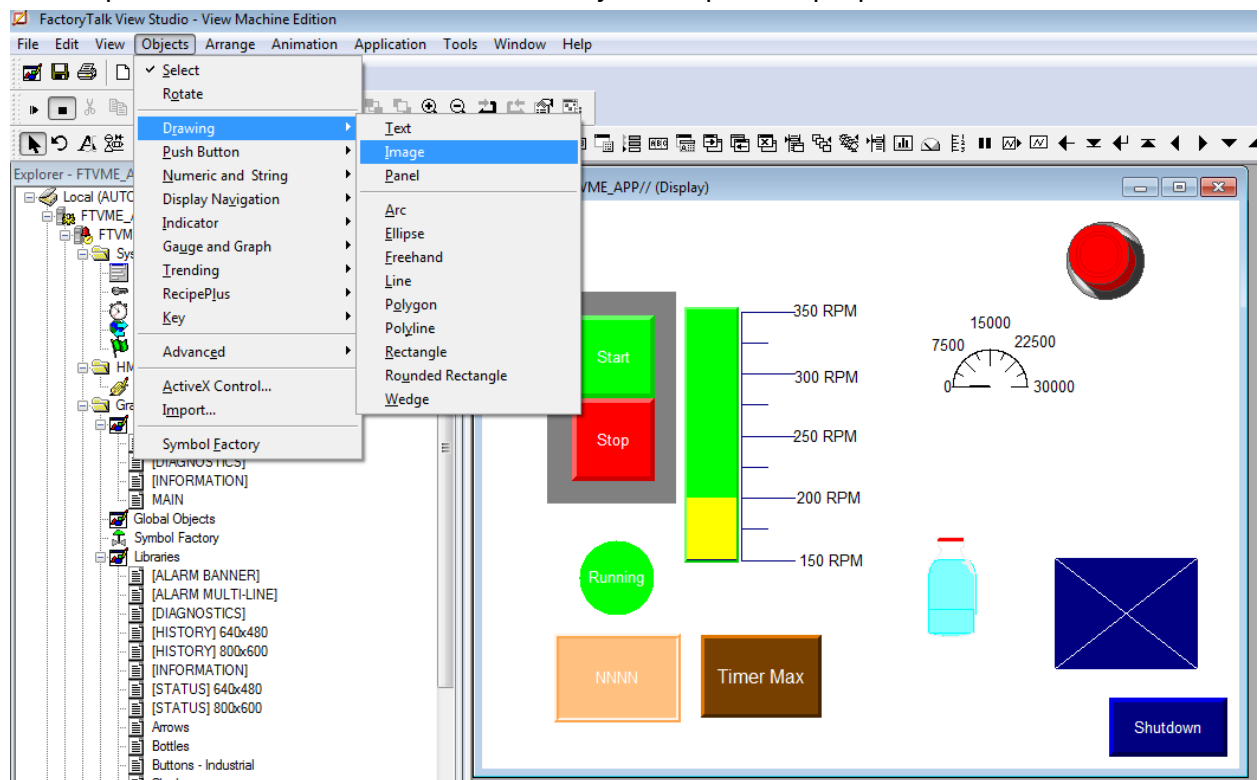




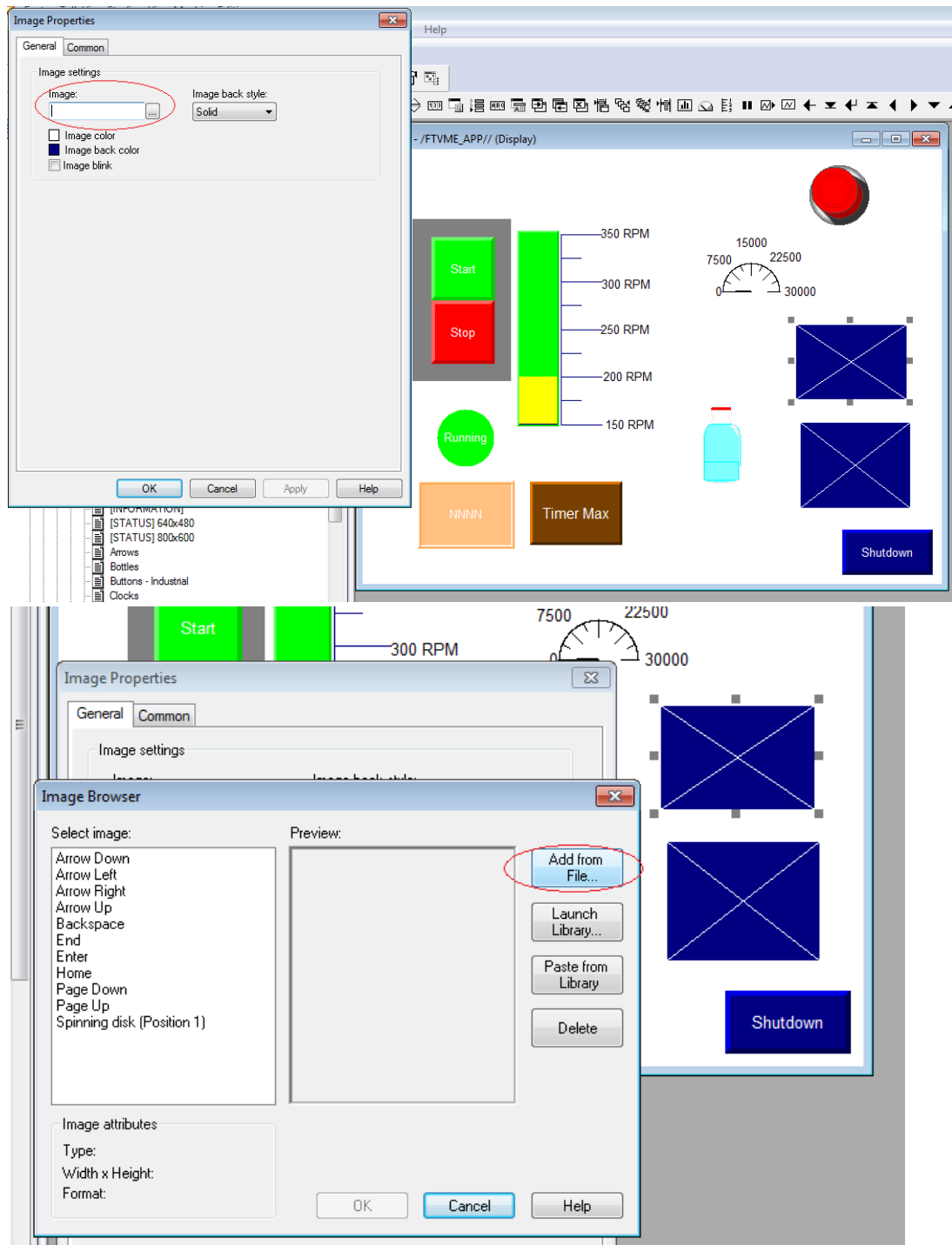
Lab15c:Importing Graphics

Note – Sometimes it is necessary to use a bitmap image (.BMP) or a JPEG image (.jpeg) on a display to represent various components of the system that the PanelView Plus is being used to control. As an example, in this part of the lab, we will add a background to the display.

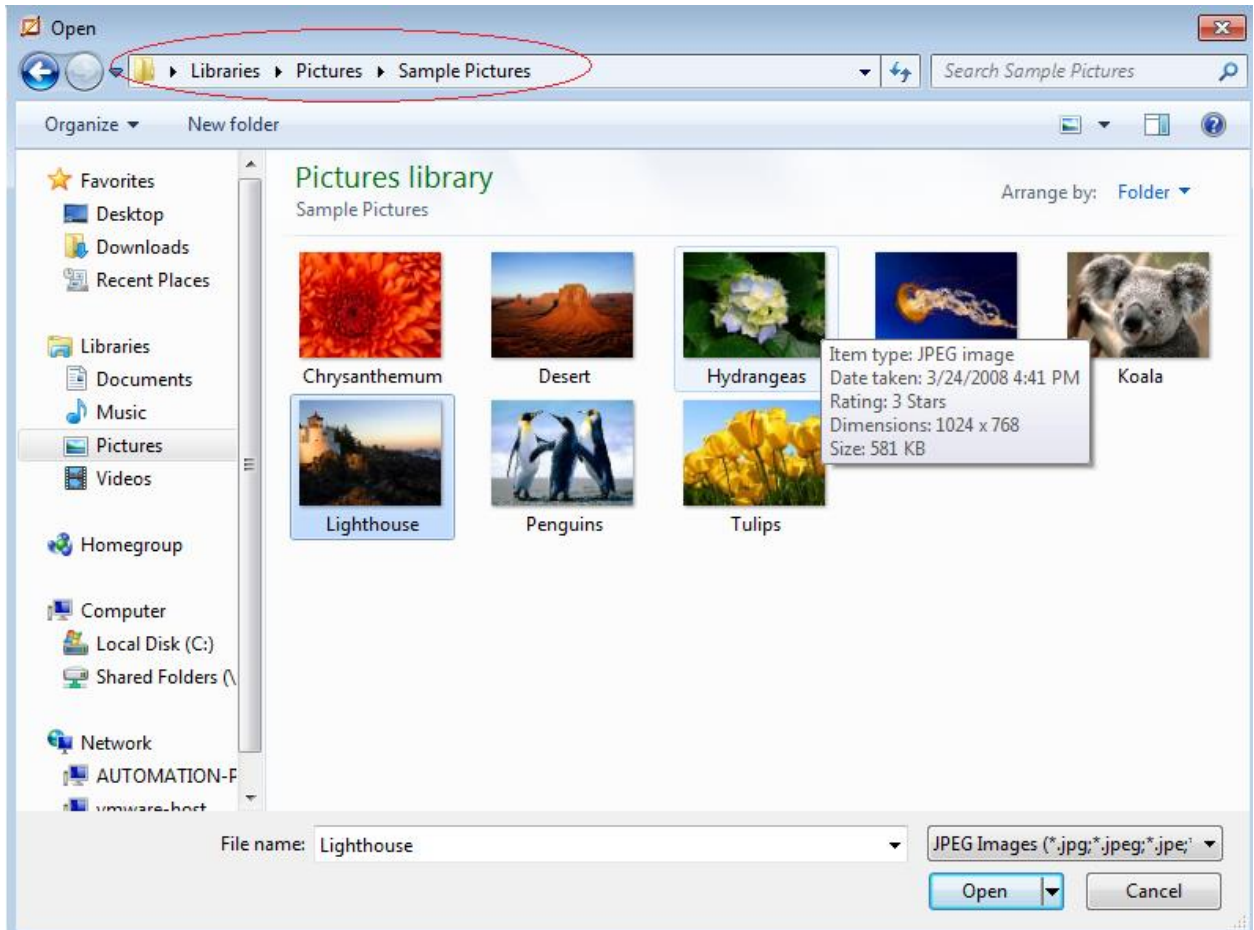
1. To use an image object, start out by going to “Objects→Drawing→Image”. Once you have selected this item, in the blank space just above the image from lab 15b, draw a basic square and then double click on that object to open the properties window.



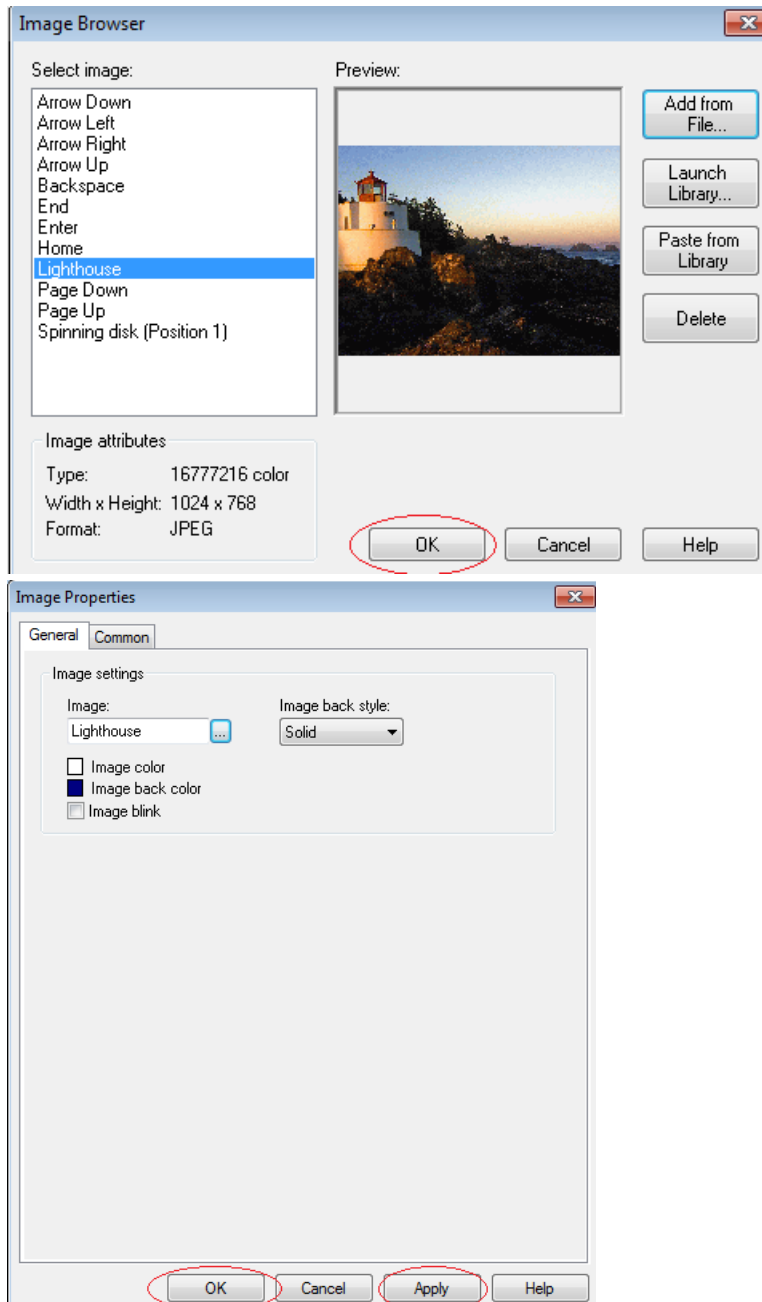
2. Once the properties window is open, we need to browse for an image. Within the image properties window, click the “browse” button. Once you click the browse button the image browser window will open. Once this window opens, we need to specify where we want to get an image from. Click the “Add from file...” button to select an image as shown below.



3. After clicking the “Add from file..” button in step 2, you will be prompted to select a location from which to choose an image. The file path for each PC may be different for where the sample pictures are located. For this lab the file path is shown below. Once you locate your sample pictures, select one and press “Open” at the bottom of the window.



4. With your image selected in the image browser, click the “OK” button at the bottom of the browser window. Click the “Apply” followed by the “OK” button on the image properties window that follows as shown below.



5. Once you are back at your main display, resize your new image so that it covers the entire display. Right click on the image and select “Arrange→Send to back”. This will have made your newly imported image the background of your MAIN display.

Lab 15 review:

1. Can you drag and drop gadgets from the symbol factory?
2. What two picture file types can you add to a display?

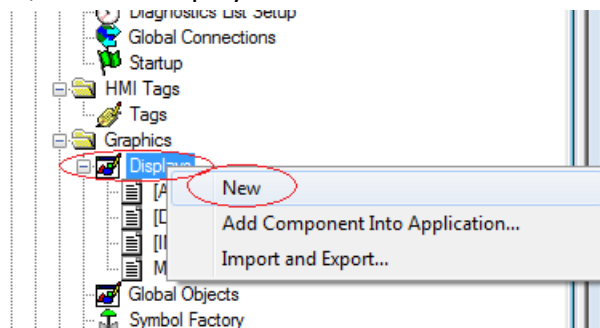


This now completes lab 15. Please wait for instructions to proceed further.

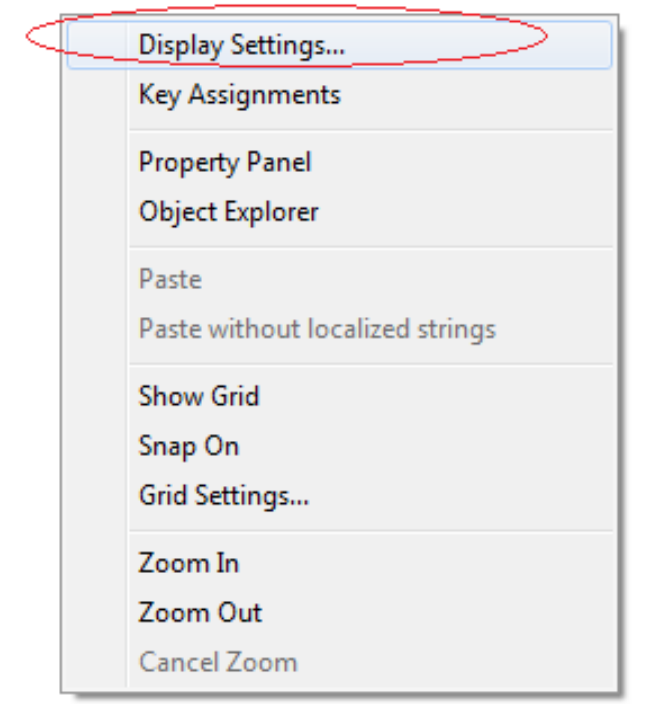
Lab 16: Changing screens

Objective: To inform the user on how to change screens within their PanelView Plus project via the display. At the end of this lab, the user will be able to move between multiple screens and back again.

1. The first thing we need to do in this lab is to create a new display to switch to. Start by right-clicking on “Displays” in the project explorer and select “New”. This will create a new, untitled display.

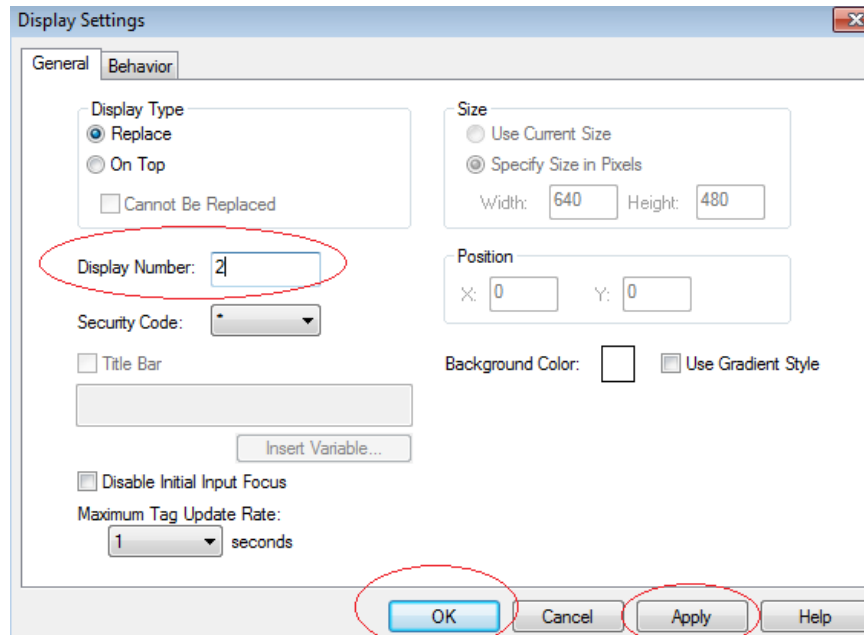


2. Within the newly created blank display, right-click and select “Display Settings”. This will open the display settings window allowing us to make changes to our new display and how it behaves.

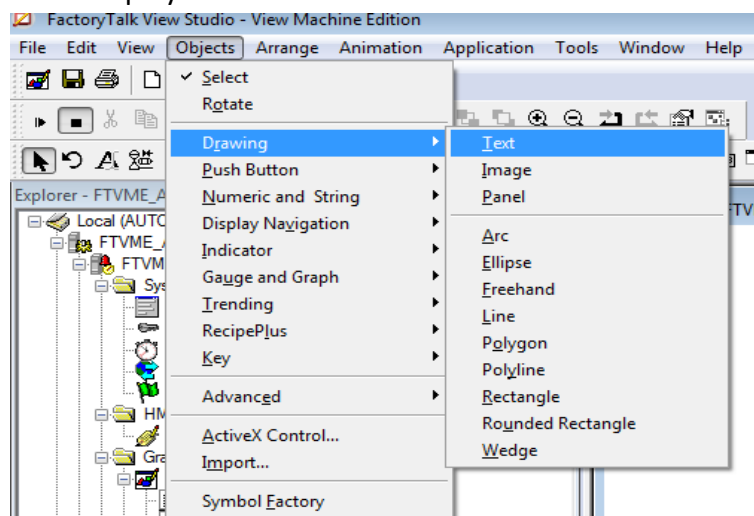


3. Once the display settings window is open, there are a few changes we need to make. Enter the following information into the “Display Settings” window:
 - Display Number: 2

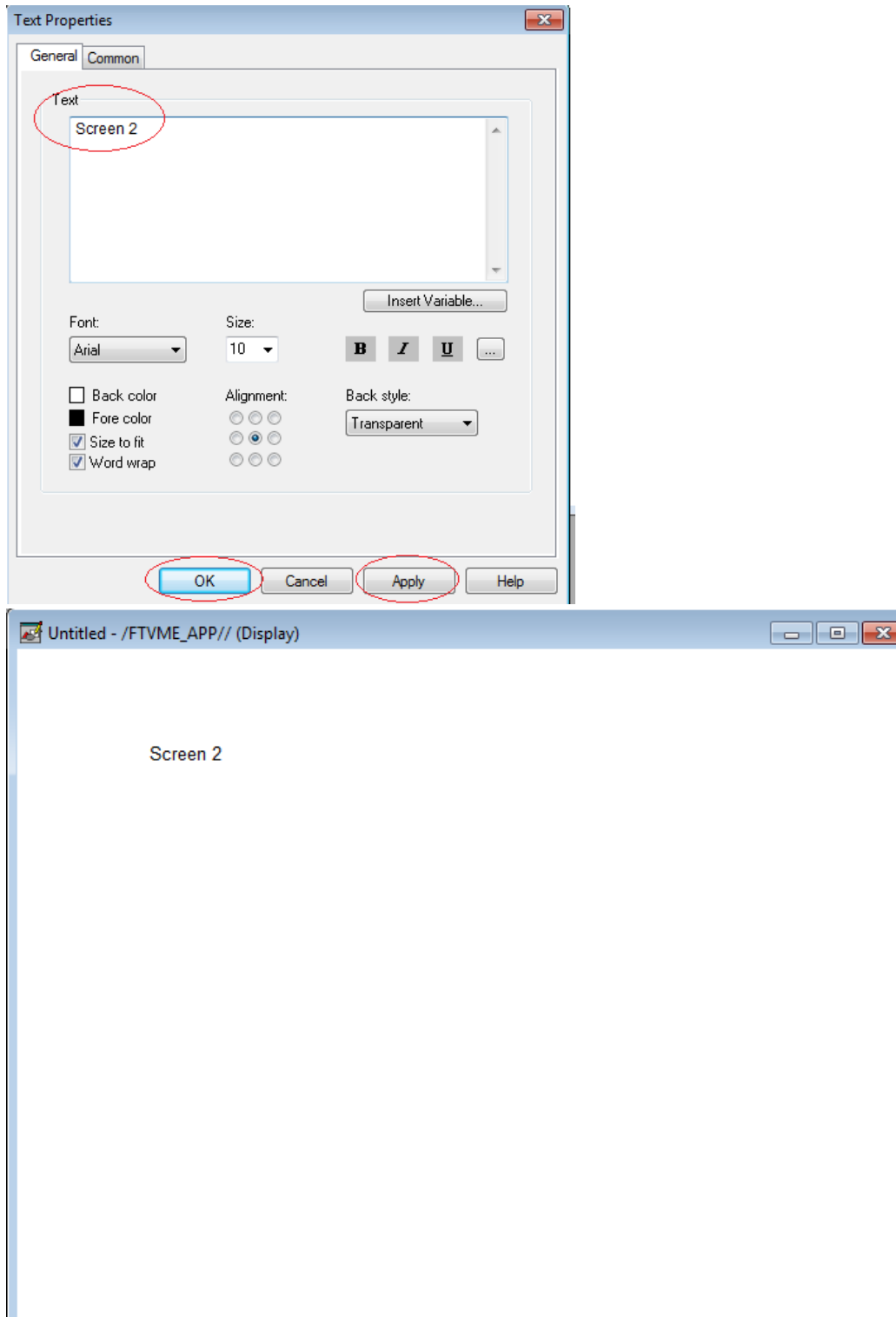
Once you have entered the following information, click the “Apply” followed by the “OK” button at the bottom of the “Display Settings” window.



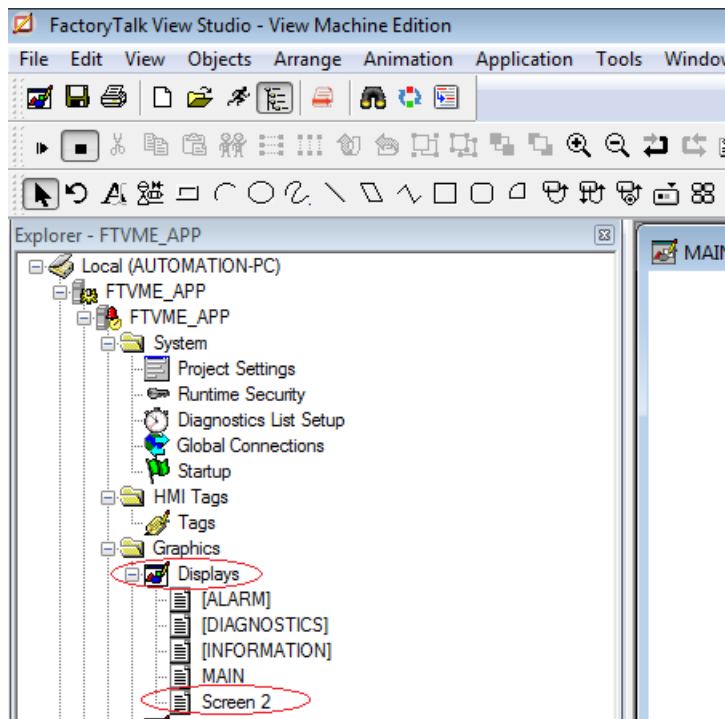
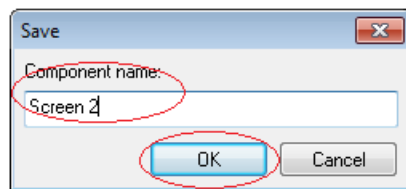
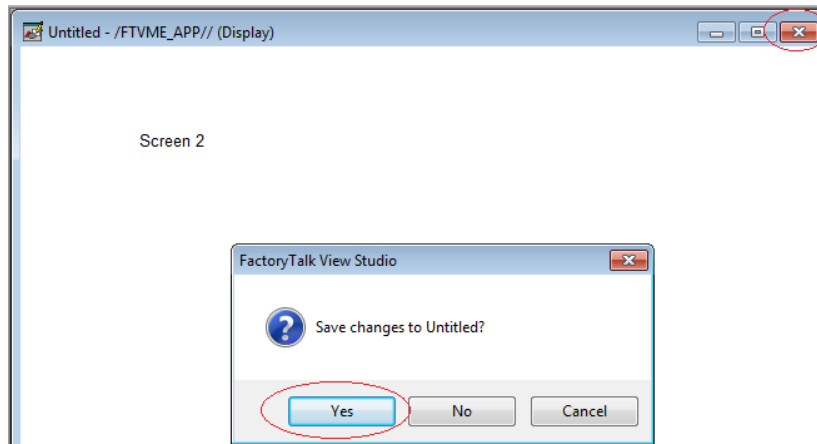
4. After clicking “Apply” and “OK” from step 3, you will be brought back to your blank display. We will add some text to it saying “Screen 2”. This is done by going to “Objects→Drawing→Text”. Once this option is selected, draw a basic text box on the blank display.



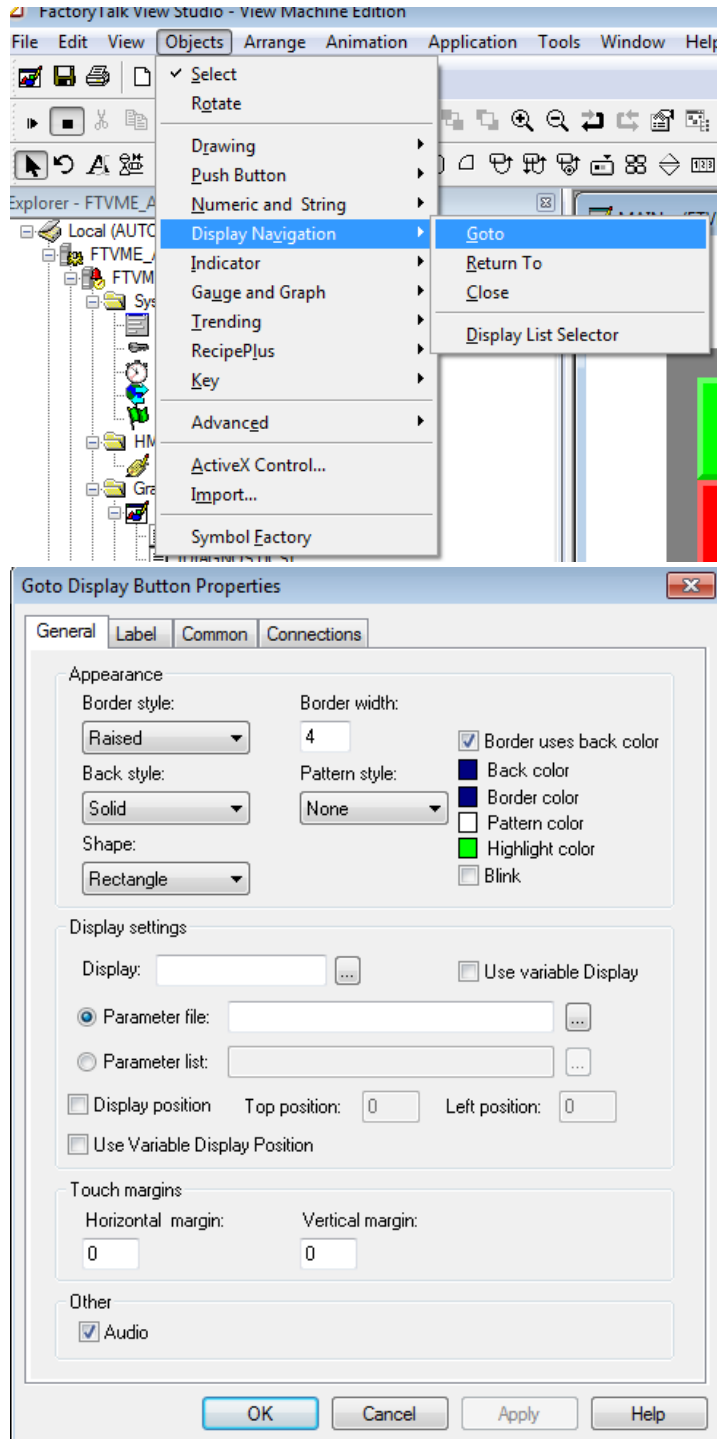
- After drawing a text box in step 4, the “Text Properties” window will open. Within this window, enter the text “Screen 2” and then press the “Apply” followed by the “OK” button at the bottom of the test properties window. Your new display should look similar to the image below.



- After making changes to our new display, we need to save our changes. This can be done by clicking the “X” in the upper right-hand corner of the new display which closes it. Before the display closes, you will be prompted to save changes. Click “Yes” to save, and when asked to name the display, name it “Screen 2” and click “OK”. You will now notice a “Screen 2” within the “Displays” list of the project explorer as shown below.



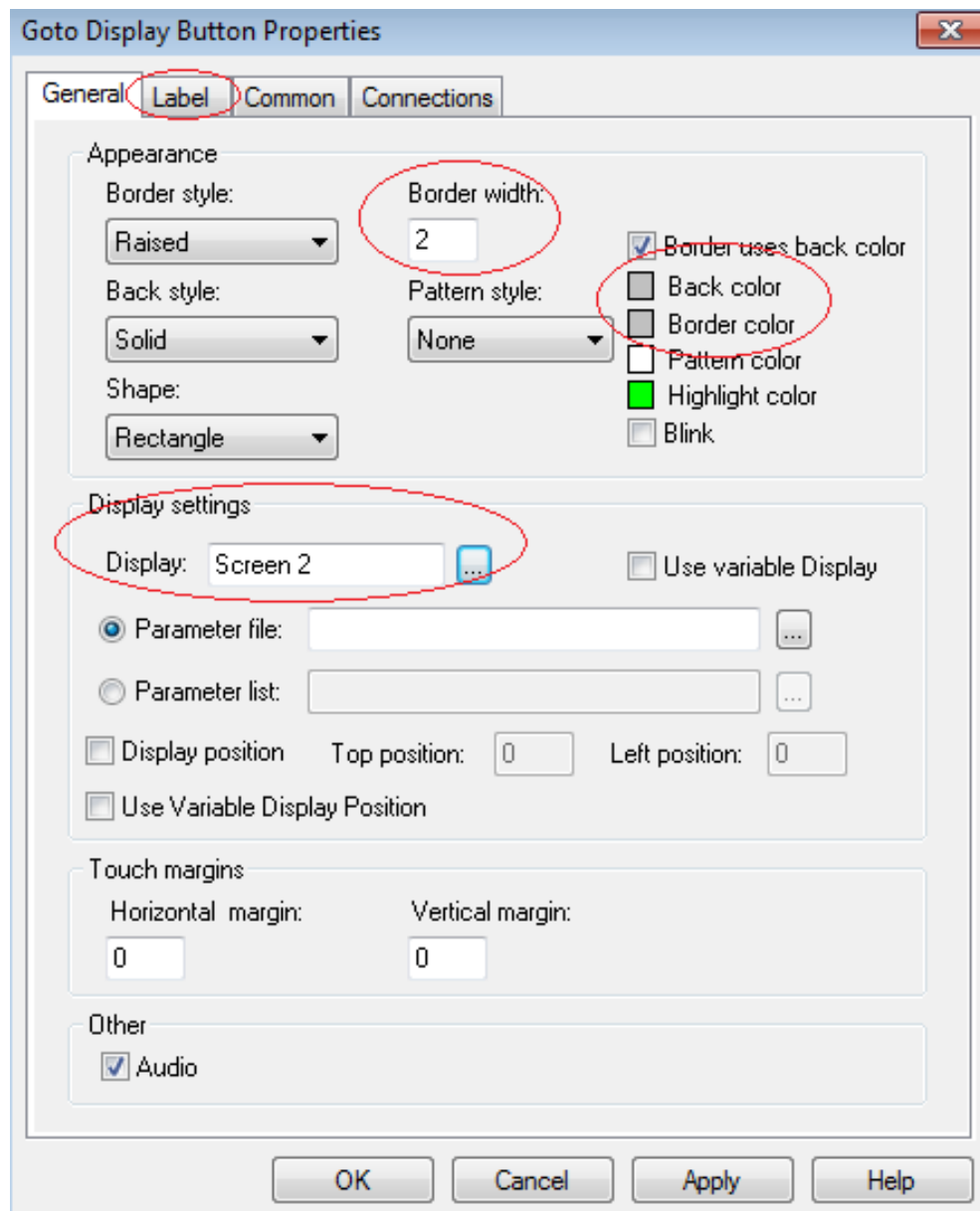
7. The next step in screen navigation is to add a “Go to” button to our displays so that we can navigate back and forth. You can find the “Go to” button by going to “Objects→Display Navigation→Goto”. Once you have selected this option, draw a basic square with your cursor on the MAIN display. This will open the Go to button display properties.



8. The first tab that will be opened when the properties window opens, is the “General” tab. We need to make several changes to this tab. Enter the following information into the fields of the general tab:

- Border Width: 2
- Back / Border Color: Light Gray
- Display: Screen 2

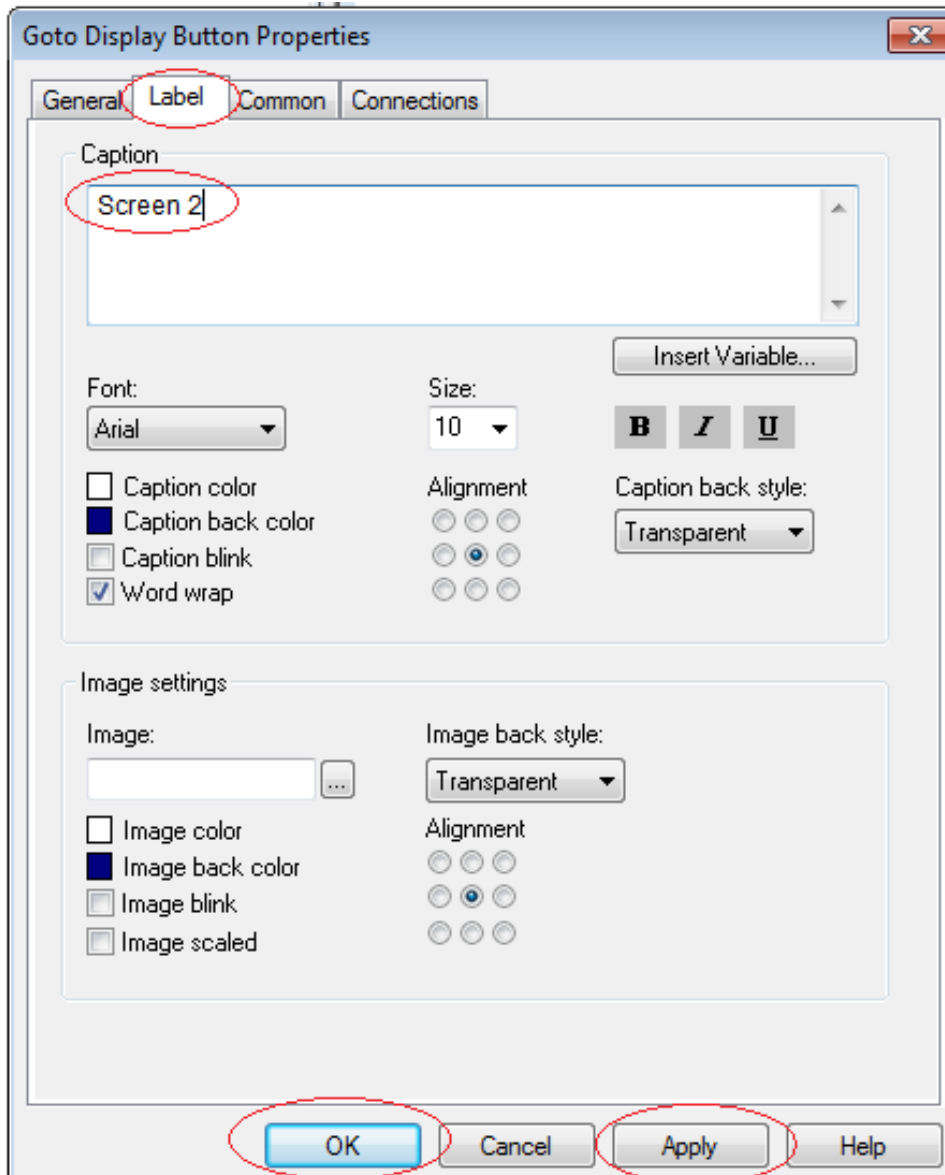
Your general tab should now look like the one in the image below. Once you have successfully made these changes, click over to the “Label” tab.



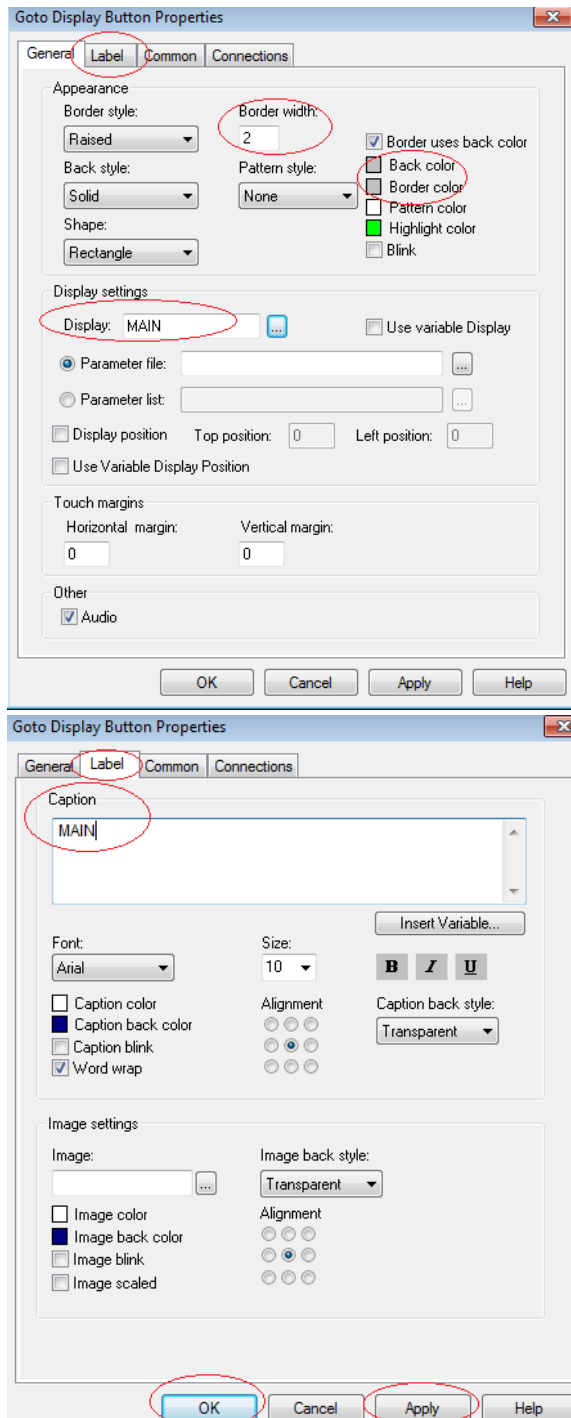
9. Once the “Label” tab is open, there are a few changes we need to make to this tab. Enter the following information into the “Label” tab of the go to button properties window:

- Caption: Screen 2

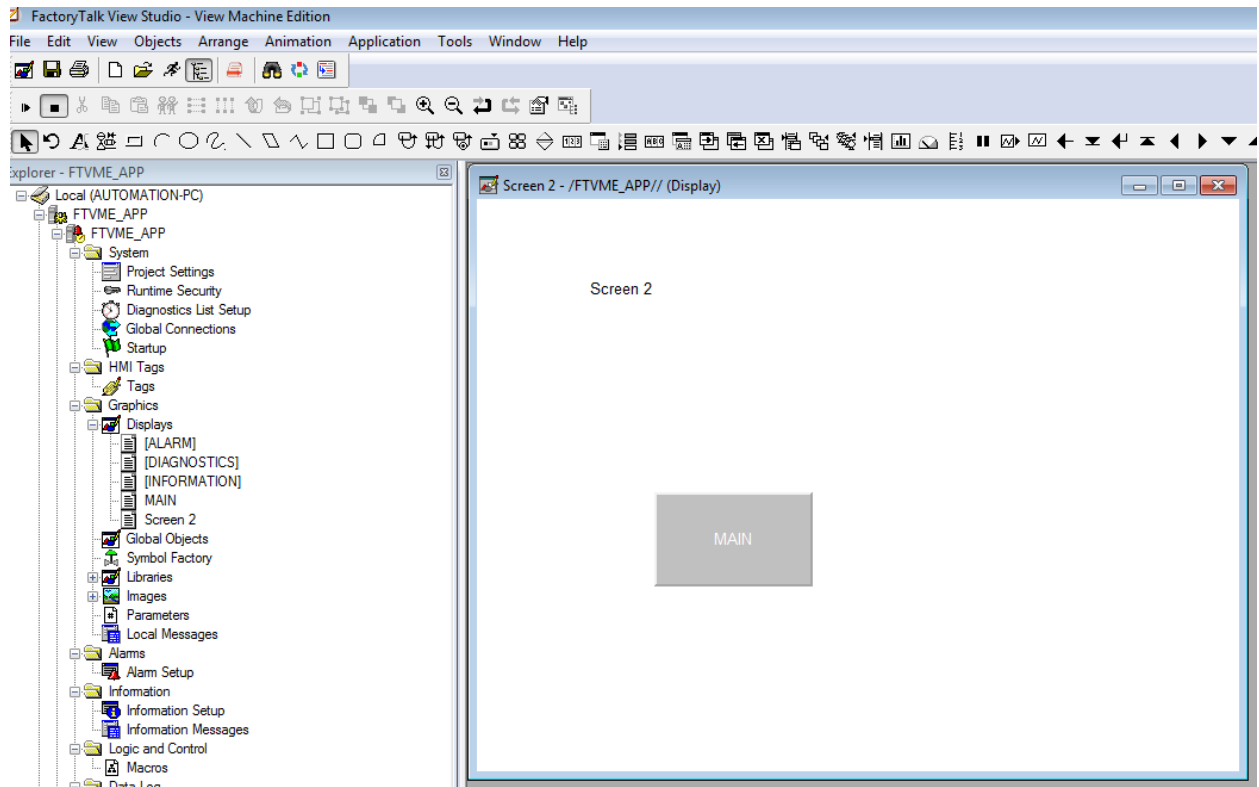
Your “Label” tab should look like the image below. Once you have successfully made these changes, click the “Apply” followed by the “OK” button at the bottom of the properties window.



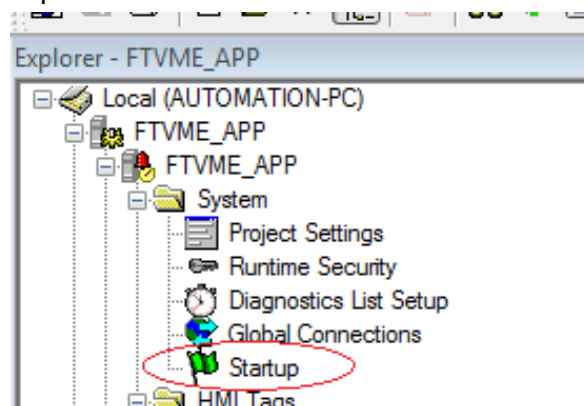
10. Now we need to create a similar go to button on our screen 2, so that we can navigate back to the MAIN display. Start by opening the display labeled “Screen 2” and drawing a go to button somewhere on the screen. Once you draw the go to button on to screen 2, the general tab of the button properties will open. Make the “general” and “label” tab look like the images below.



11. After editing the properties of the go to button on screen 2, your screen 2 display should look like the image below.



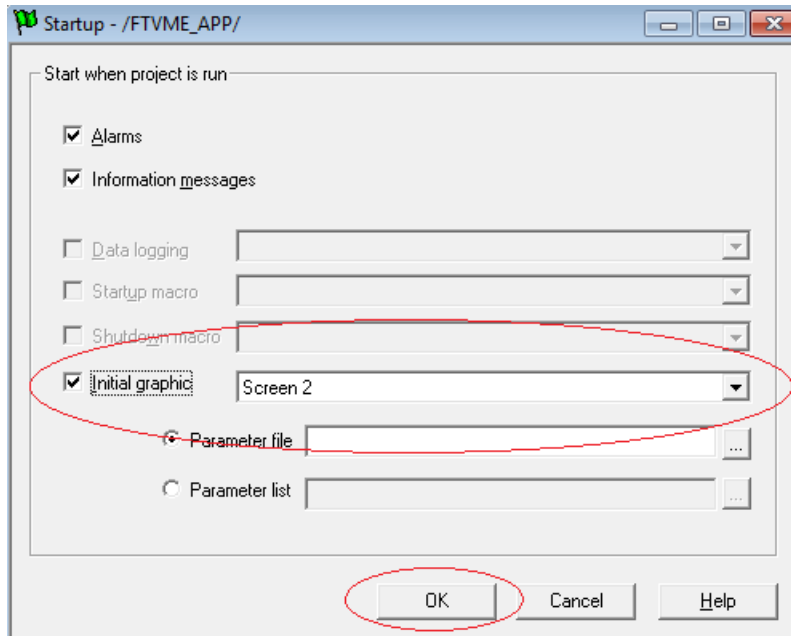
12. Now that we have created an additional screen, we need to specify which one opens first at startup. This can be configured by double clicking on "Startup" within the project explorer.



13. Once you double-click “Startup” from step 12, enter in the following information to the start-up window:

-Initial Graphic: Screen 2

Your startup window should look like the image below. Once you have made the changes successfully, click the “OK” button at the bottom of the start-up window.



14. Now that we have specified which screen will be open at start-up, it's time to save our application, create a new runtime application, download to the panelview and test our navigation buttons. If you can't remember this process, take a look at lab 11. If you get stuck, please seek the help of one of the instructors.

Lab 16 Review:

1. In the go to button properties, what tab do you identify the display you want to go to?
2. Where do you select the initial graphic for the program?
3. How do you create a new display?
4. How do you rename a display?
5. When “Replace” is selected as the “Display Type” what happens to the open display?

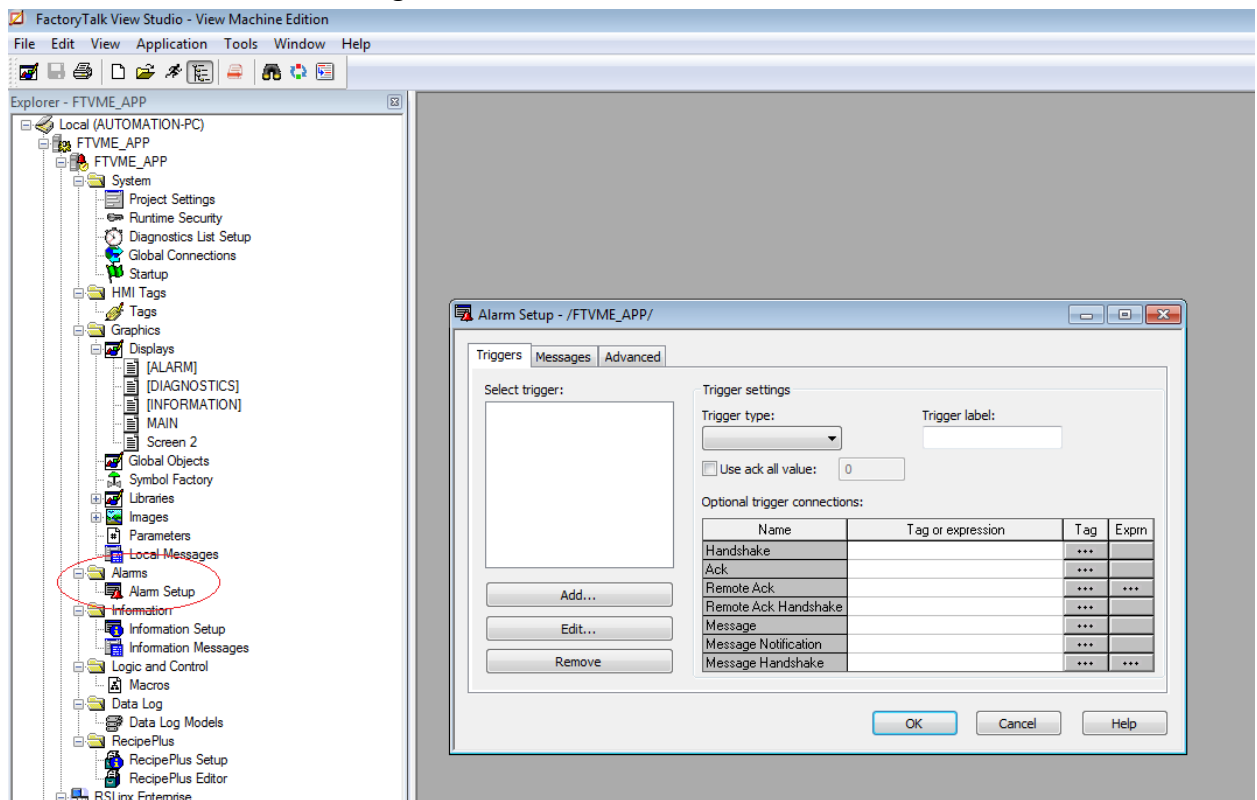


This now completes lab 16. Please wait for instructions to proceed further.

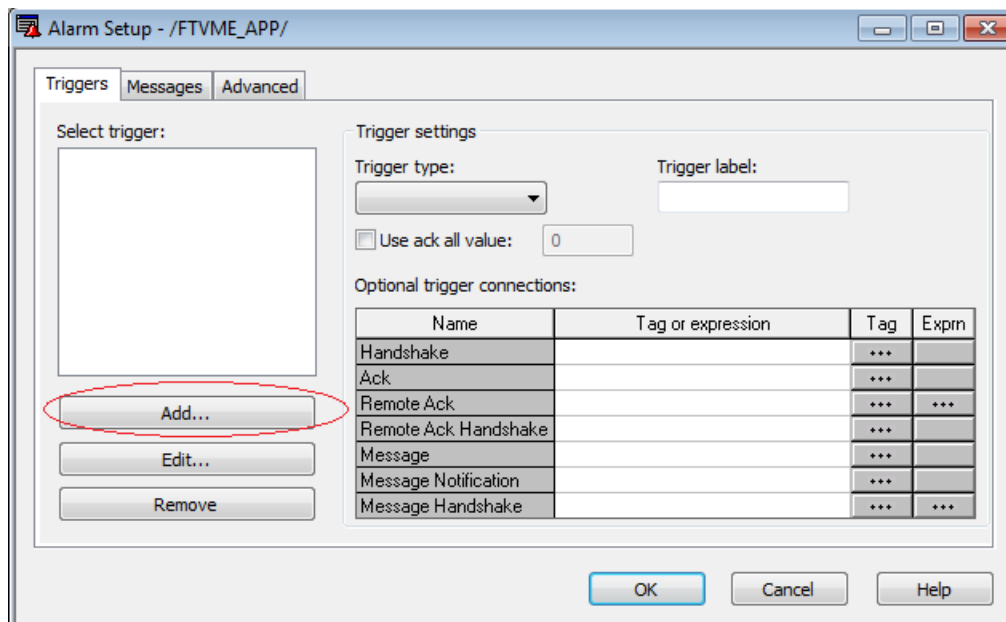
Advanced lab 1: Alarms

Objective: To guide the user through creation and configuration of alarms to be used to notify the operator of various issues within the system. In this lab, we will cover alerting the operator when the motor speed is too high. By the end of this lab, the users shall be able to make alarms for important on-the-job system information.

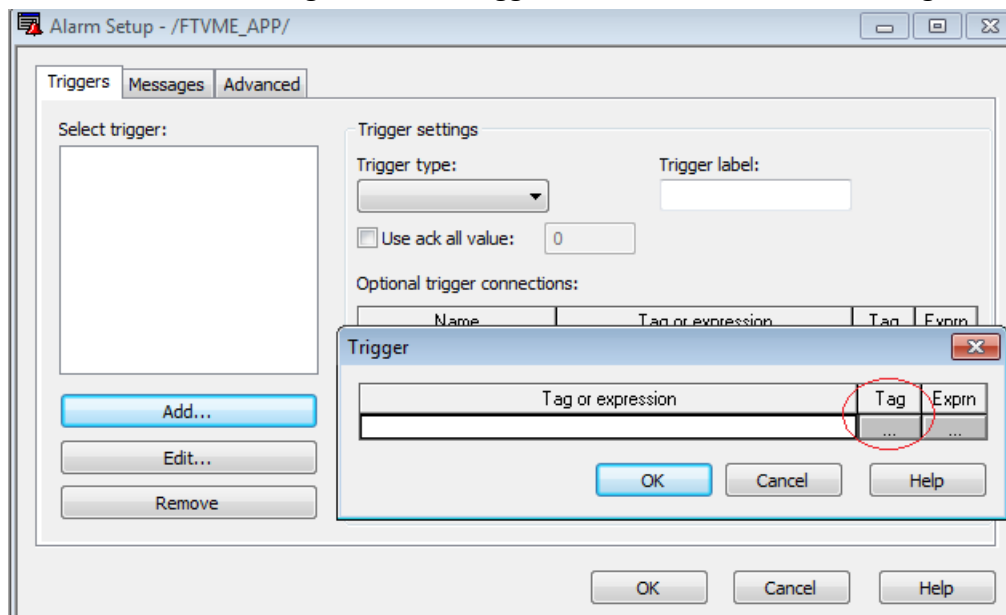
1. The first step in setting up alarms within FactoryTalk View studio is to open up the “Alarm Setup” window by clicking on the “Alarm Setup” link within the project explorer like the one shown in the image below.



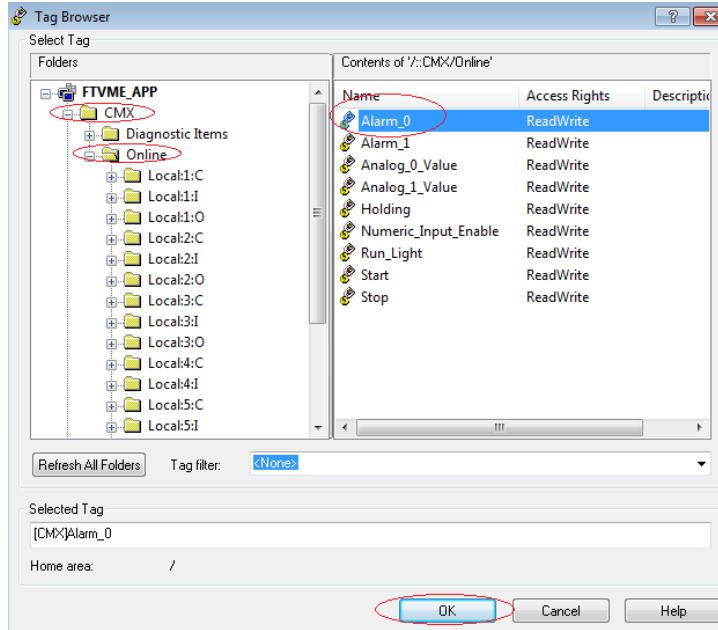
2. After clicking on “Alarm Setup” in step 1, we need to select a tag for triggering an alarm. This can be done by clicking on the “Add...” button underneath the select trigger window as shown below.



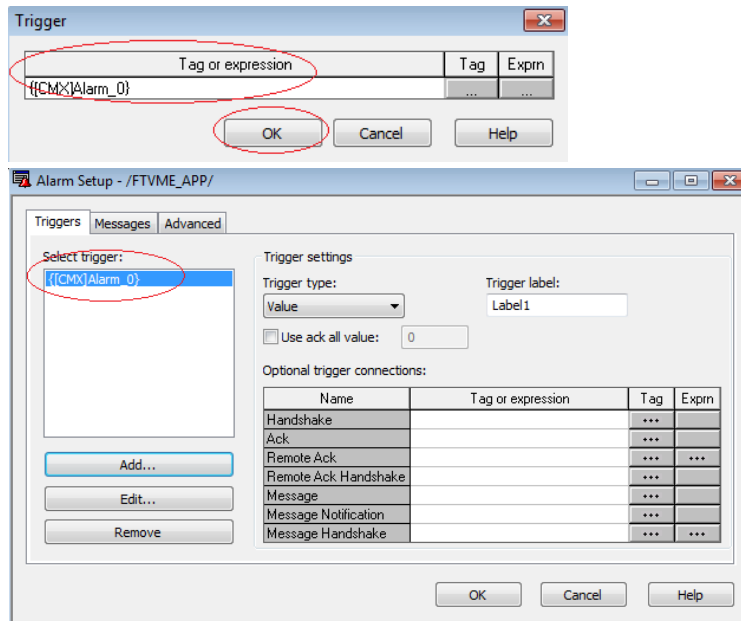
3. After clicking the “Add...” button in the previous step, a trigger window will open so that we can browse for a tag. From this trigger window, click the browse tag button.



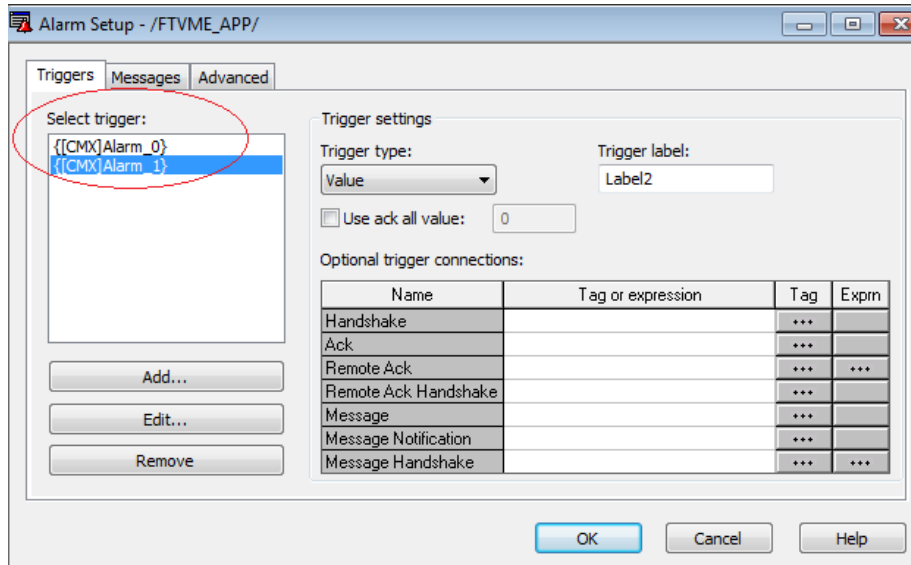
4. From the tag browser window, find the “Alarm_0” tag. This can be found by going to “CMX→Online→Alarm_0”. Once you have this tag highlighted, click the “OK” button at the bottom of the tag browser window. This will assign the tag “Alarm_0” to our alarm trigger from the previous step.



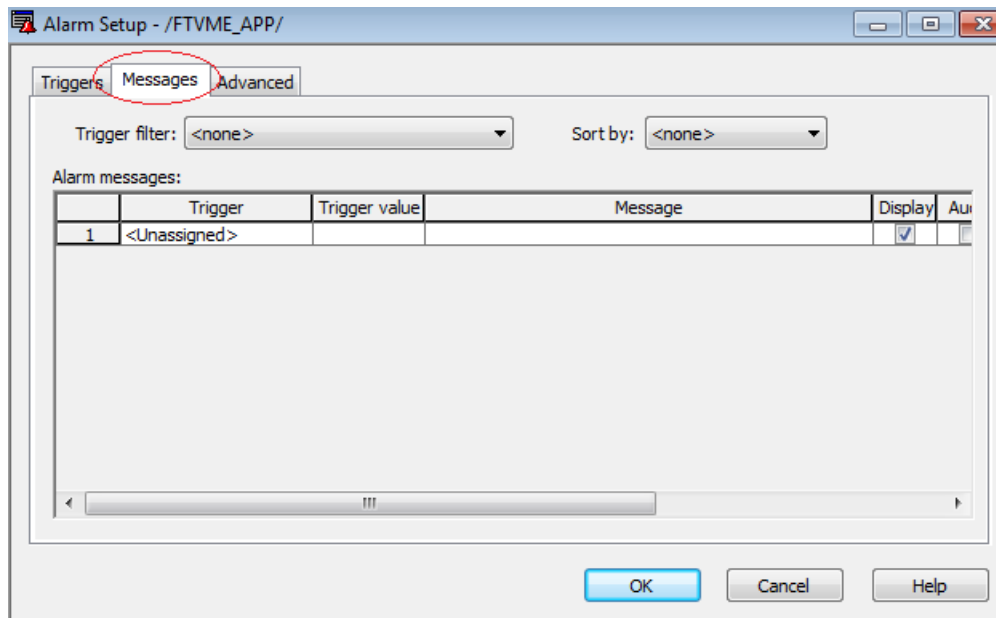
5. From the trigger window, you should see the tag “Alarm_0” within the tag or expression field. Press the “OK” button and you will be brought back to the alarm setup window with your newly configured trigger.



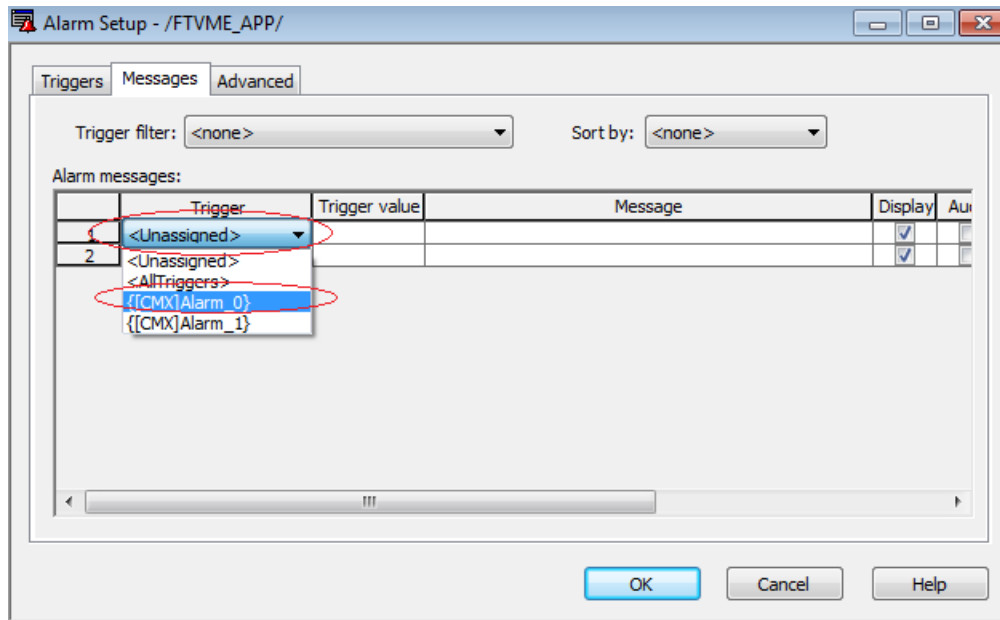
6. Now that we have added one tag as an alarm trigger, we need to add one more. Repeat steps 2 through 5 of this lab except this time choose tag “Alarm_1” from your PLC. After you have added the second tag, your alarm setup window should look like the image below. If you get stuck, please seek the help of one of the instructors.



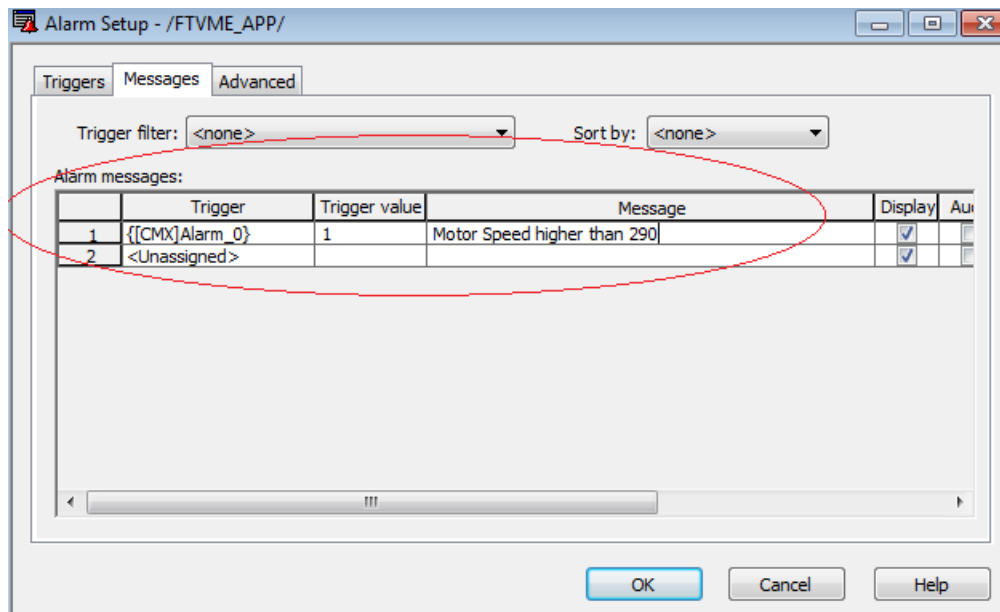
7. The next step we need to take in configuring our alarms is to click over to the messages tab so that we can define the message displayed when an alarm condition is present. Start by clicking over to the “Messages” tab of the alarm setup window as shown below.



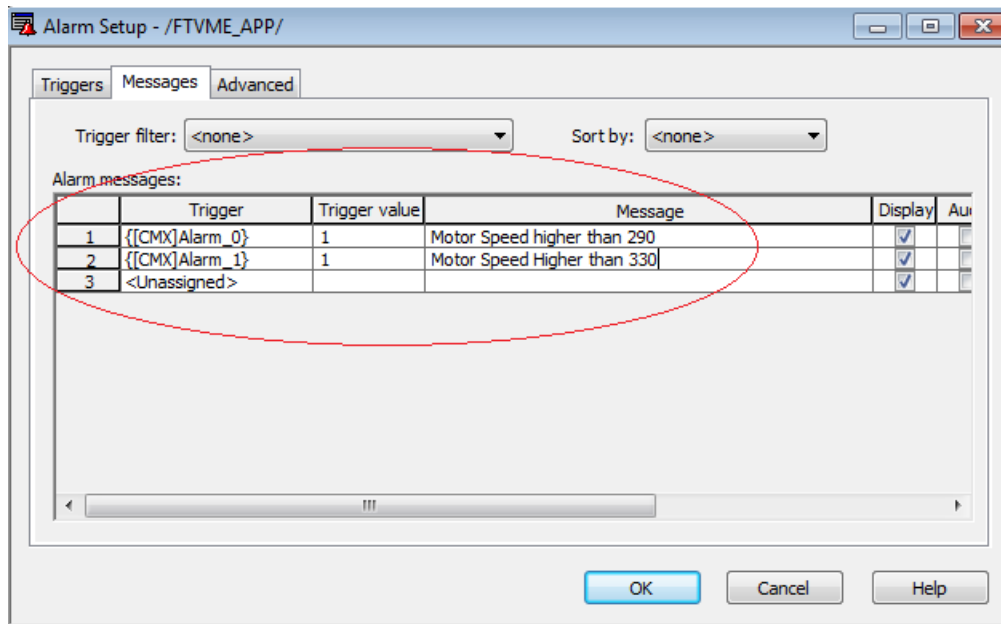
8. From the messages tab, there are a few changes we need to make. Start by clicking in the field underneath “trigger” where it says “Unassigned”. There will be a drop-down menu and from this menu select “Alarm_0”.



9. Once you have selected “Alarm_0” set the trigger value to 1, and type “Motor Speed Higher than 290” in the “Message” field. Your alarm setup window should end up looking like the image below.



10. Repeat steps 8 & 9 to add alarm 1 into the alarm messages table. Your alarm setup window should look like the image below. Once you have made these changes successfully, click the “OK” button at the bottom of the alarm setup window.



11. Now that we have configured a couple alarms, it is time to test our application on the PanelView. Save the changes you have made, create a new .MER file, and download to the PanelView for testing. Once running on the PanelView, you should be able to rotate the potentiometers until they reach a certain value that will trigger an alarm state. If you don't remember how to do all this, use lab 11 to get you through the download process. If you get stuck, please seek the help of the instructors.

Advanced Lab 1 Review:

1. Inside the alarm setup window, where do you add tags that will trigger an alarm?
2. By default, what also appears with your message inside the alarm banner?
3. What happens when the alarm bit goes high and then goes low before the alarm banner is closed or the alarm is acknowledged.
4. What happens to an alarm if another occurs after it? Does it still display?

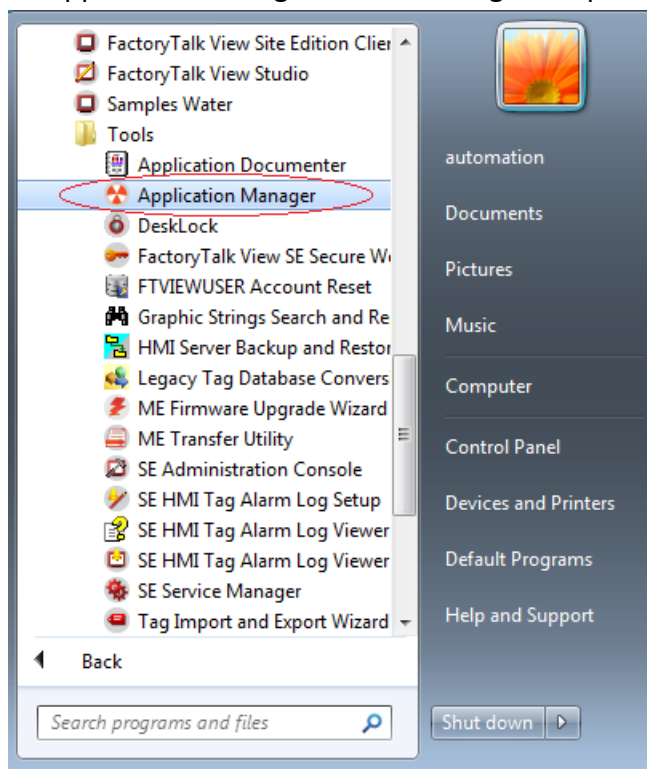


This completes advanced lab 1. Please wait for instructions to proceed further.

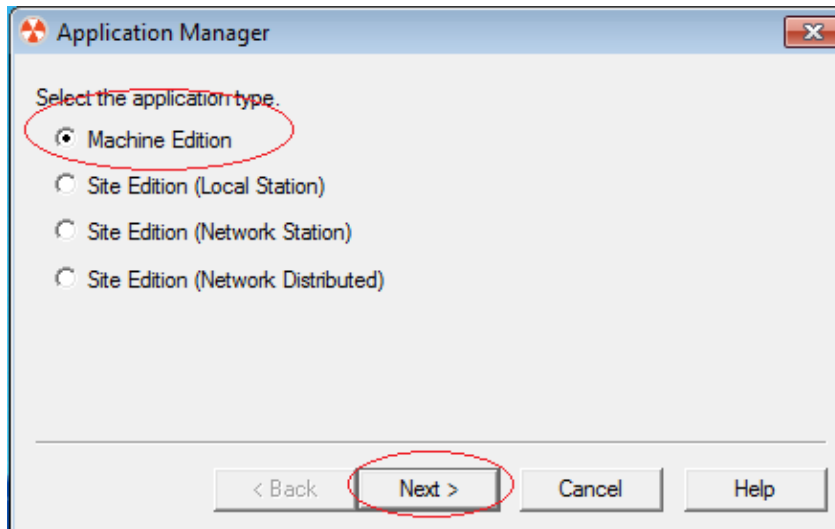
Advanced lab 2: Application Backup and Restore

With the “Application Manager” you can back up and restore applications. When an application is backed up, a file is created with a .apa file extension. Normally, all of the application files are stored within FactoryTalk View Studio’s root directories. Backing up a file allows you to store an application in a folder of your choice to be used later. A backed up file can also be emailed and restored on a different computer. When a runtime file is created, a .MER file is produced. It is important to realize that this file can be transferred to a different computer, but will open in FactoryTalk ME Station instead of FactoryTalk View Studio, allowing the runtime to be viewed, but not edited.

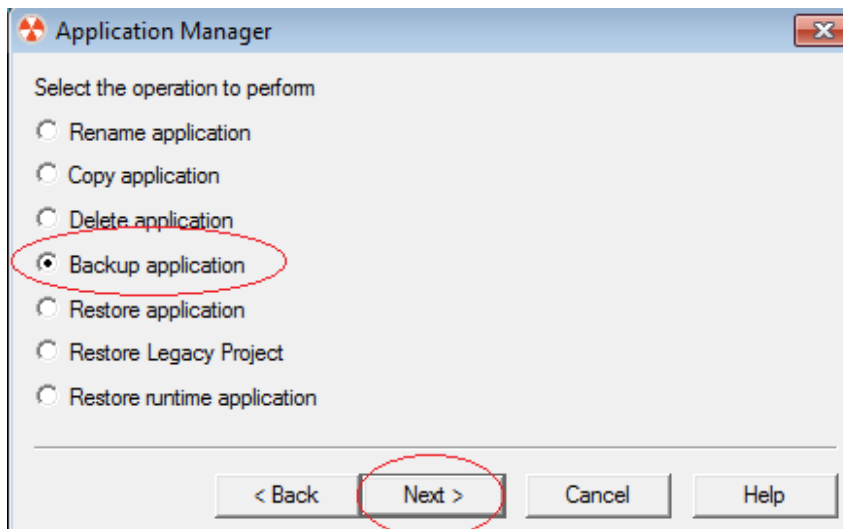
1. The first step in backing up an application like the one we have developed in this course is to open up the “Application Manager”. You can find this by going to “Start→All Programs→Rockwell Software→Factory Talk View→Tools→Application Manager”. Click on application manager and it will begin to open.



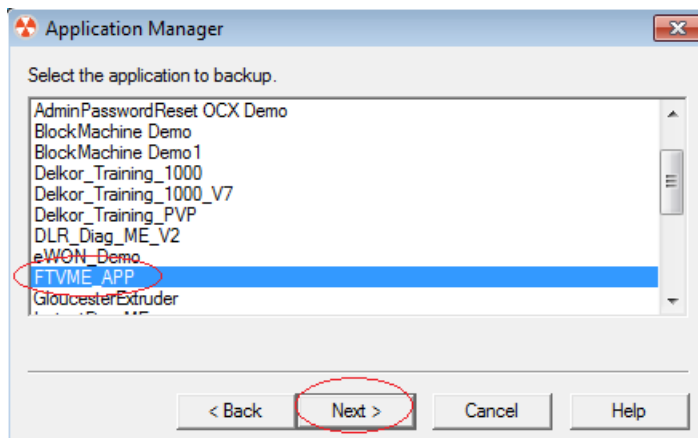
2. Once the Application Manager is open, the initial screen will have you choose which programming environment the application resides in. Select the “Machine Edition” option followed by the “Next” button at the bottom of the window.



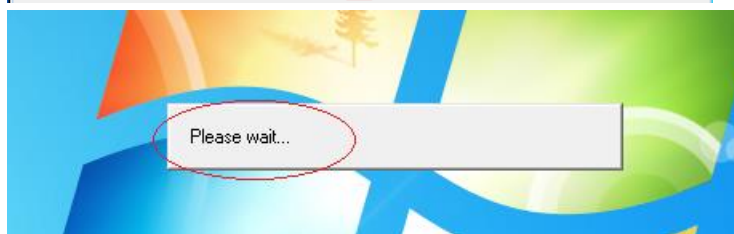
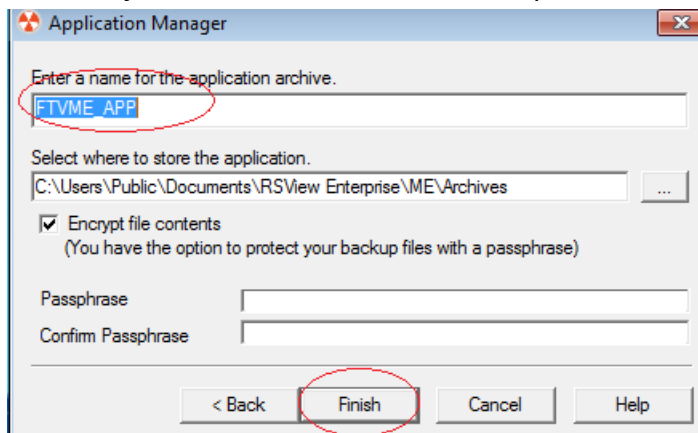
3. The next thing you will be asked is which action to take (EG: Rename / copy / delete / etc.). Choose the “Backup Application” option followed by the “Next” button at the bottom of the window.



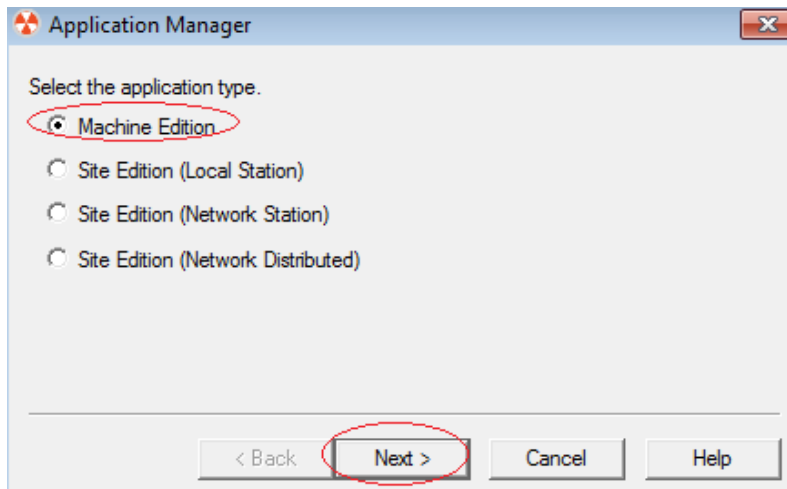
- After clicking “Next” from step 3, you will be asked to choose an application to back-up from a list of existing machine edition applications. Choose the “FTVME_APP” from the list, followed by the “Next” button at the bottom of the window.



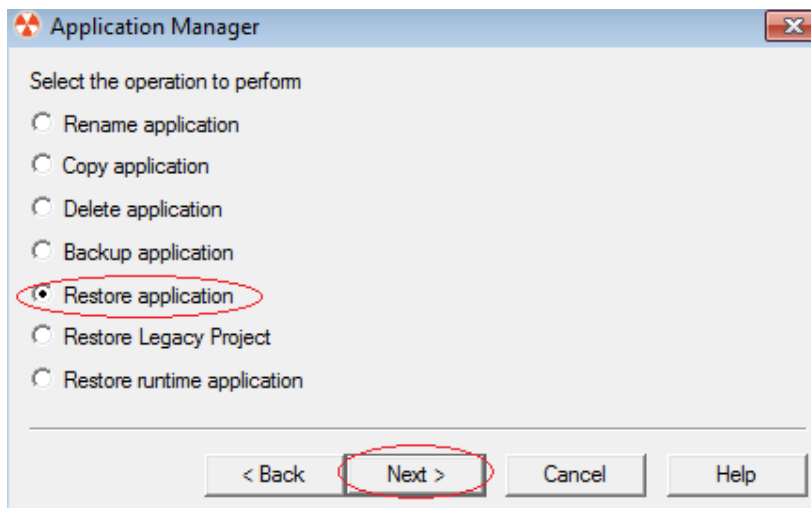
- The final step before the back-up process begins is to give your .APA file a name. In this case we will leave the default name of “FTVME_APP” in the name field and click the “Finish” button at the bottom of the window. This will begin the back-up process, during which time you will see a “Please Wait” message appear on your desktop. After this message goes away, the process is complete. There is no message telling you that it finished, just know that it is done at this point.



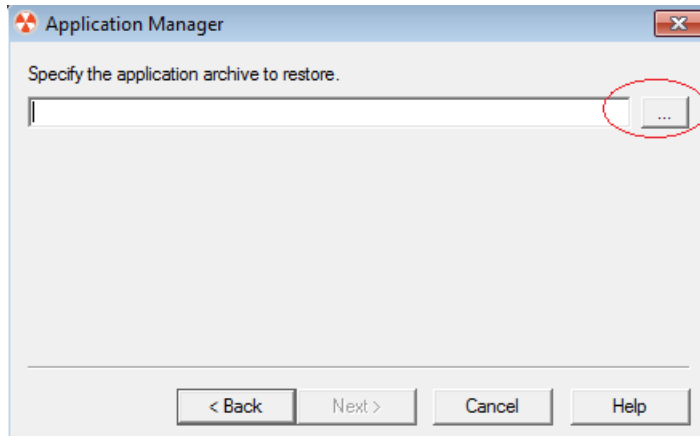
6. Now that we have explored the process of backing up an application, we can take a look at the process of restoring an application. Start by opening up application manager again from “Start→All Programs→Rockwell Software→ Factory Talk View→Tools→Application manager”. Once application manager opens, choose the “Machine Edition” option followed by the “Next” button at the bottom of the window.



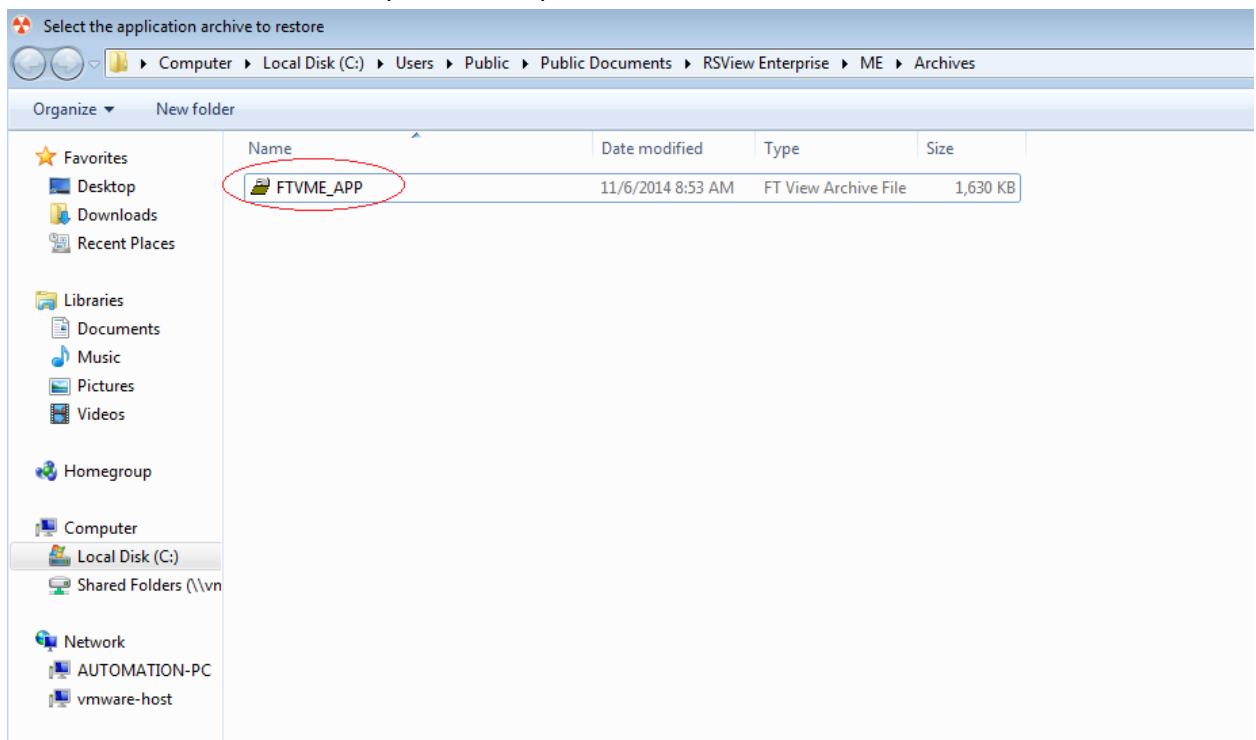
7. The next step is to choose the action we want to take. In this case, choose the “Restore Application” option followed by the “Next” button at the bottom of the window.



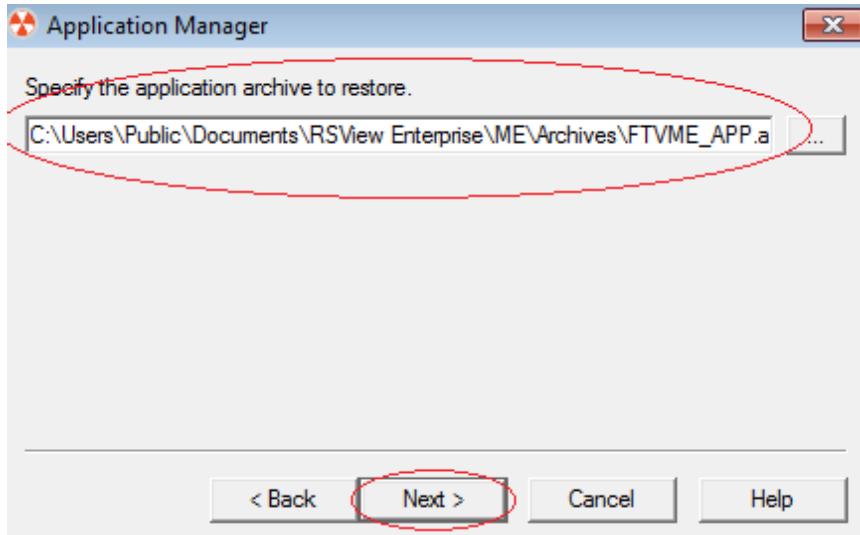
8. After clicking “Next” from step 7, you will be asked to browse to a .APA file that you want to restore. Start by clicking the browse button as shown below.



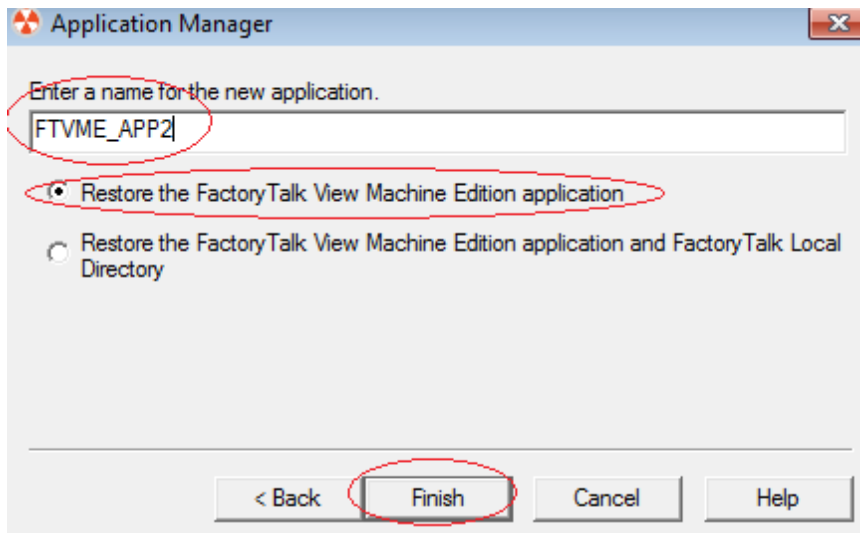
9. Select the “FTVME_APP” .APA file from the list of .APA files followed by the “Open” button at the bottom right corner of the window. Note – the reason this file is here is because we made the back-up of it in steps 1 – 5.



10. After clicking “Open” from step 9, you will be brought back to the application manager window. You should now see the file path and name of your application in the “Specify the application archive to restore” field. Click the “Next” button to proceed.



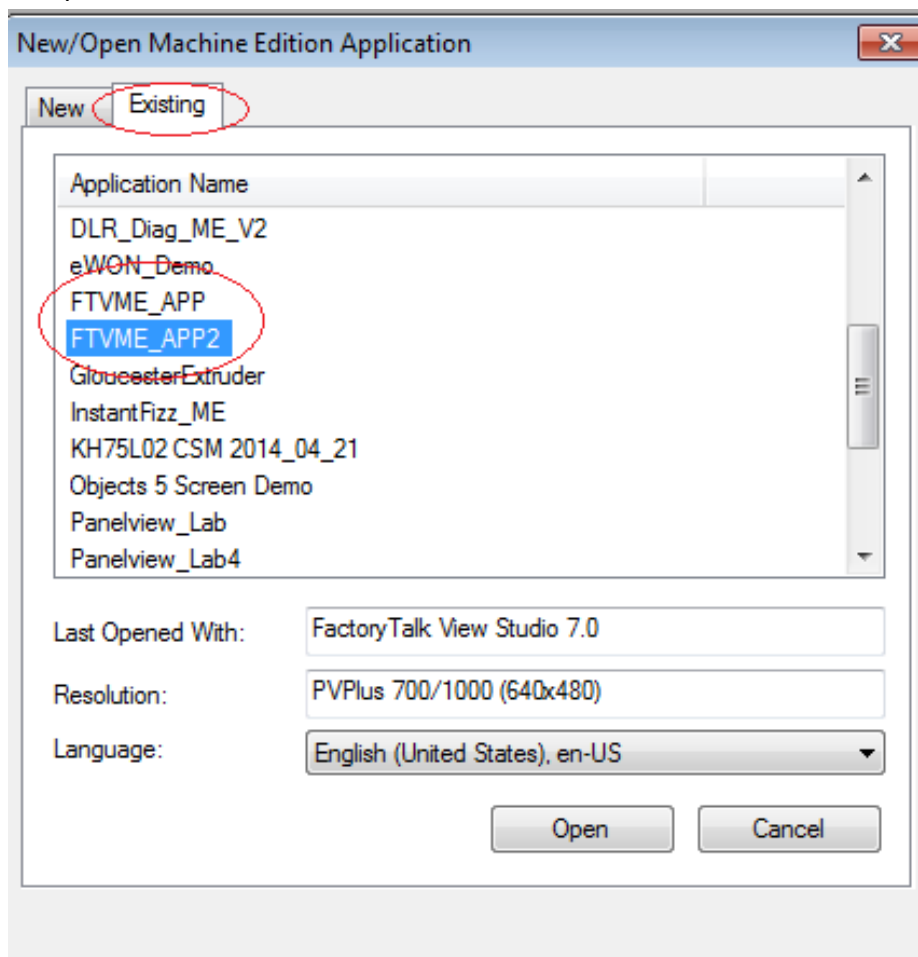
11. After clicking “Next” from the previous step, you may get a message stating that the file was passphrase protected and that it needs the passphrase to continue. If this is the case, simply leave the passphrase field blank and click to proceed further. You should now be at a point where the application manager is asking you to name the new application. In this case, since the application “FTVME_APP” already exists, we will need to specify a different name. Give the new application the name of “FTVME_APP2”, leave the option of “Restore the FactoryTalk View Machine Edition application” checked, and click the “Finish” button at the bottom of the window.



12. After clicking “Finish” from step 11, there will be a message on your desktop saying “Please Wait”. During this time, the application we specified in step 11 is being restored. Once this message goes away, the process is completed.



13. The next step is to verify that our application was restored into Factory Talk View Studio. We will do this by opening up Factory Talk View Studio, selecting Machine Edition and then looking through the list of existing applications for the name “FTVME_APP2”. If your new / open machine edition application window looks like the image below, you have successfully backed up and restored an application and this lab is complete.



Advanced Lab 2 Review:

1. When an application is backed up, what is the new file extension?
2. What is the benefit of creating a backup file?
3. Can you save the backup file in any location or is it locked to a defined folder?
4. When restoring a backup file, can you create a new name for the file?

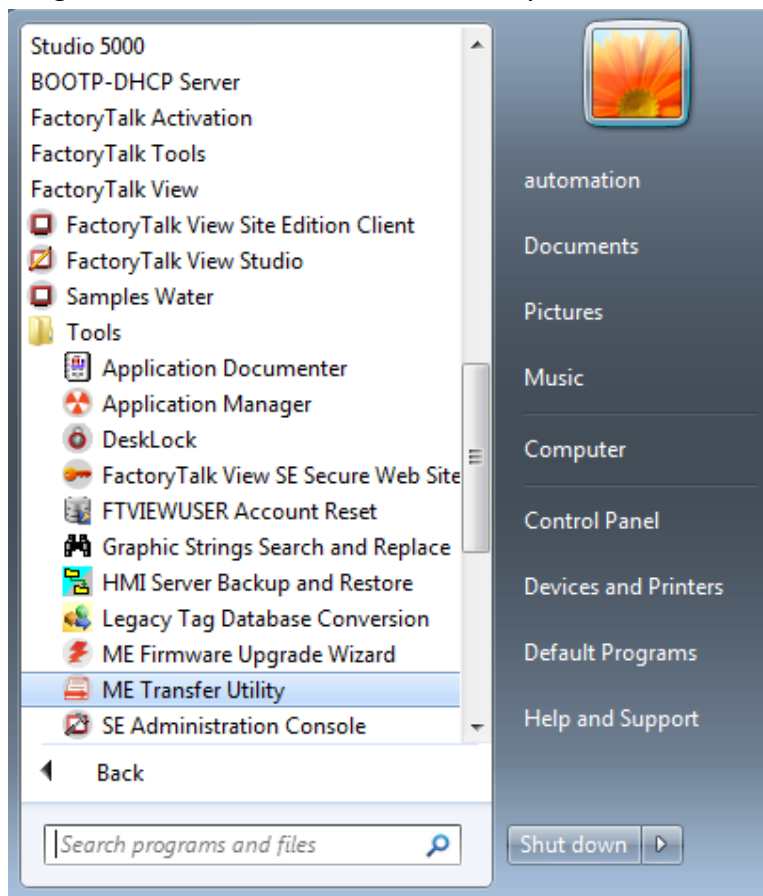


This now completes the advanced lab 2. Please wait for instructions to proceed further.

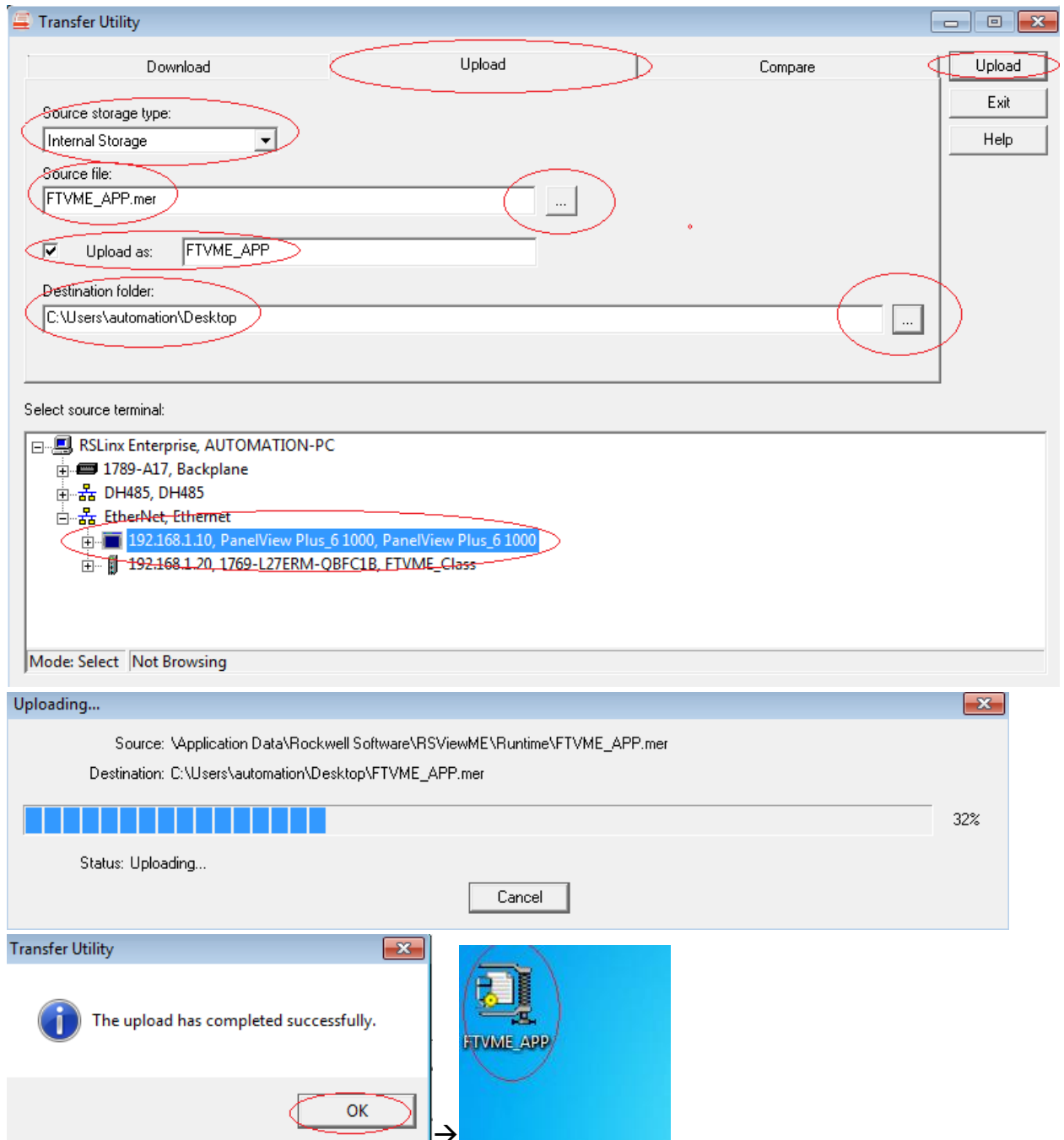
Advanced Lab 3: Uploading a program

In terms of a PanelView (source) and a computer (destination) the process of uploading is described as transferring a runtime file (.MER) from the source to the destination. This file type is only used by a PanelView or ME Station and cannot be used in FactoryTalk View Studio. Later in this manual, you will convert the runtime (.MER) file to a design time file that can be used with FactoryTalk View Studio for program edits.

1. We will start by opening up the ME Transfer Utility by going to “Start→All Programs→Rockwell Software→ Factory Talk View→ Tools→ ME Transfer Utility”.



2. Once the ME transfer utility is open, click over to the “Upload” tab. We need to specify that we are uploading from the PanelView’s internal storage, browse for the “FTVME_APP” and choose the desktop of our PC as the destination for the upload. Your Transfer utility window should look like the image below. Once you have made these changes, highlight the IP address of the PanelView in the “Source Terminal” window, and click the “Upload” button in the upper right corner of the Transfer utility window.



3. If your upload was successful, you will see a new icon of your desktop shown in the images on the previous page labeled as “FTVME_APP”. This is the .MER file that was uploaded from the PanelView. If you have this icon on your desktop, this lab is completed.

Advanced Lab 3 Review:

1. What is the file extension for a runtime file?
2. What three options are available under the source storage type?
3. If the “upload as” box is unchecked, what will be the filename when it is saved in the destination?
4. What are the two uses of the runtime file?

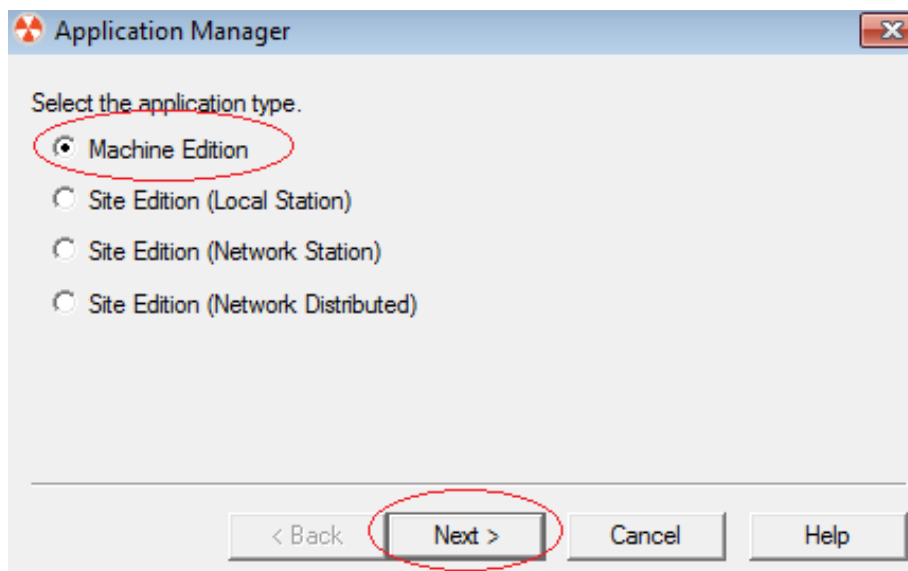


This now completes advanced lab 3. Please wait for instructions to proceed further.

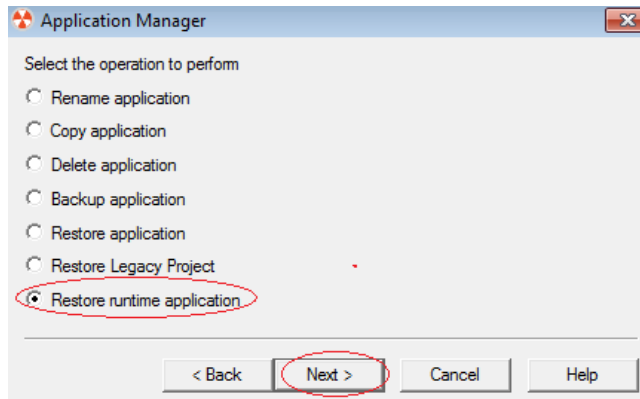
Advanced lab 4: Restoring Runtime Applications

FactoryTalk View Studio version 5.0 and later allows the user to restore a runtime application (.MER) back to a development application file (.MED) with three stipulations. During the process of creating the runtime file the user has the option of permanently protecting the file against conversion, or password protecting the file. If the file is password unknown or the option of permanently protecting the file was selected, the file cannot be restored. Additionally the runtime file has to be created at firmware 5.0 or later.

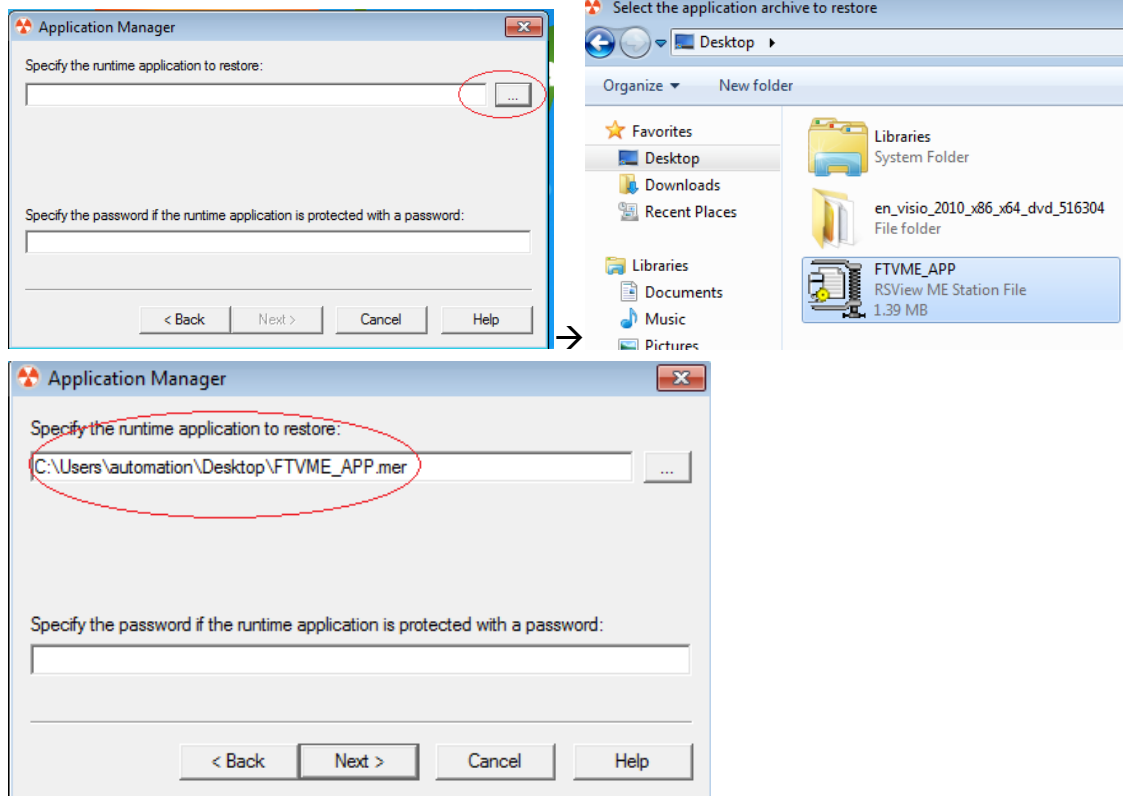
1. In this lab we will work to restore the .MER file that we just uploaded from the previous lab. Start by opening up the application manager. Once the application manager is open, select the “Machine Edition” option followed by the “Next” button at the bottom of the window.



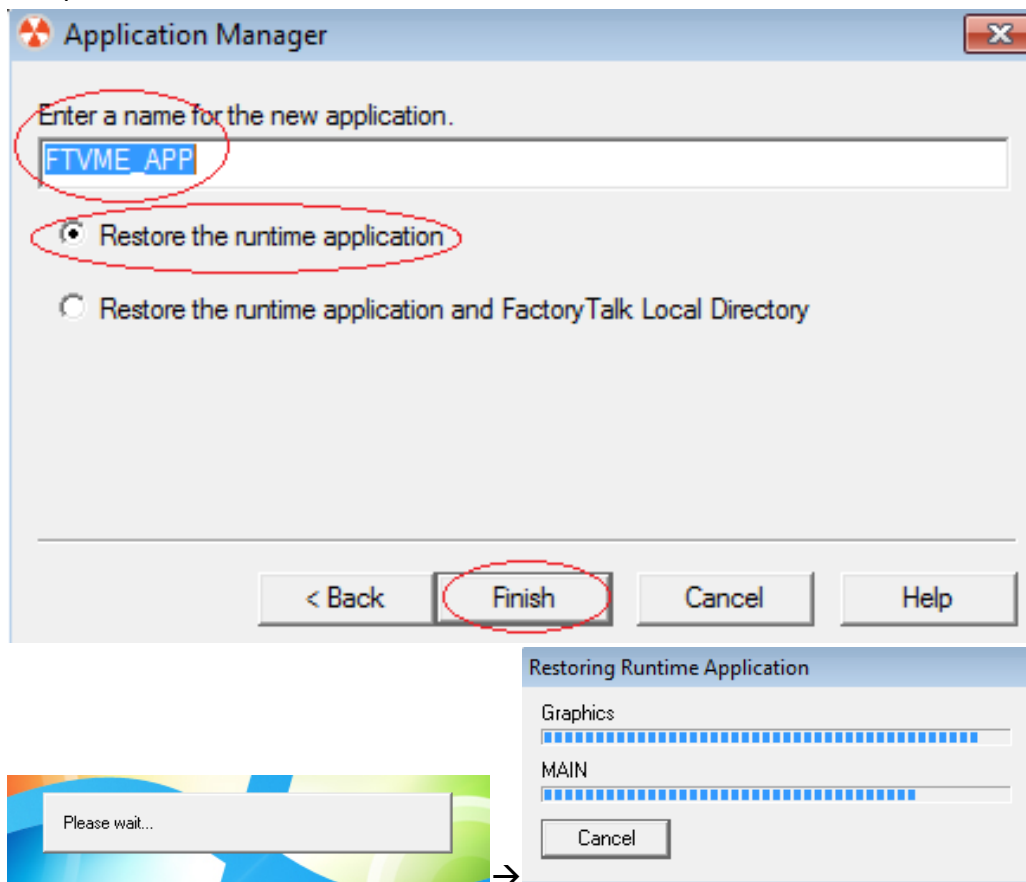
- The next step we need to take is to select which action we would like to take with the application manager. In this case, choose “Restore Runtime Application” followed by the “Next” button.



- The next step will be to browse to the location of where the .MER file is that we want to restore. Click the “Browse” button, browse to your desktop, select the .MER file named “FTVME_APP”, click open at the bottom of the window, and you will be brought back to the application manager window with the file path and name of the application we are about to restore. Click the “Next” button at the bottom of the window.



- After clicking “Next” from step 3, you will be asked to enter a name for the new application. In this case since we have already used the names “FTVME_APP & FTVME_APP2”, we will name this new one “FTVME_APP3”. Leave the default option of “Restore Runtime Application” checked and click the “Finish” button at the bottom of the window. During the restore process, you will see a message on your desktop saying “Please Wait” followed by a loading bar. Once these go away, the restore process is complete.



Advanced Lab 4 Review:

1. Name two of the three reasons why you cannot restore a runtime file?
2. What version of FactoryTalk View Studio and later allows the user to restore runtime applications?
3. What is the purpose of restoring a runtime file?

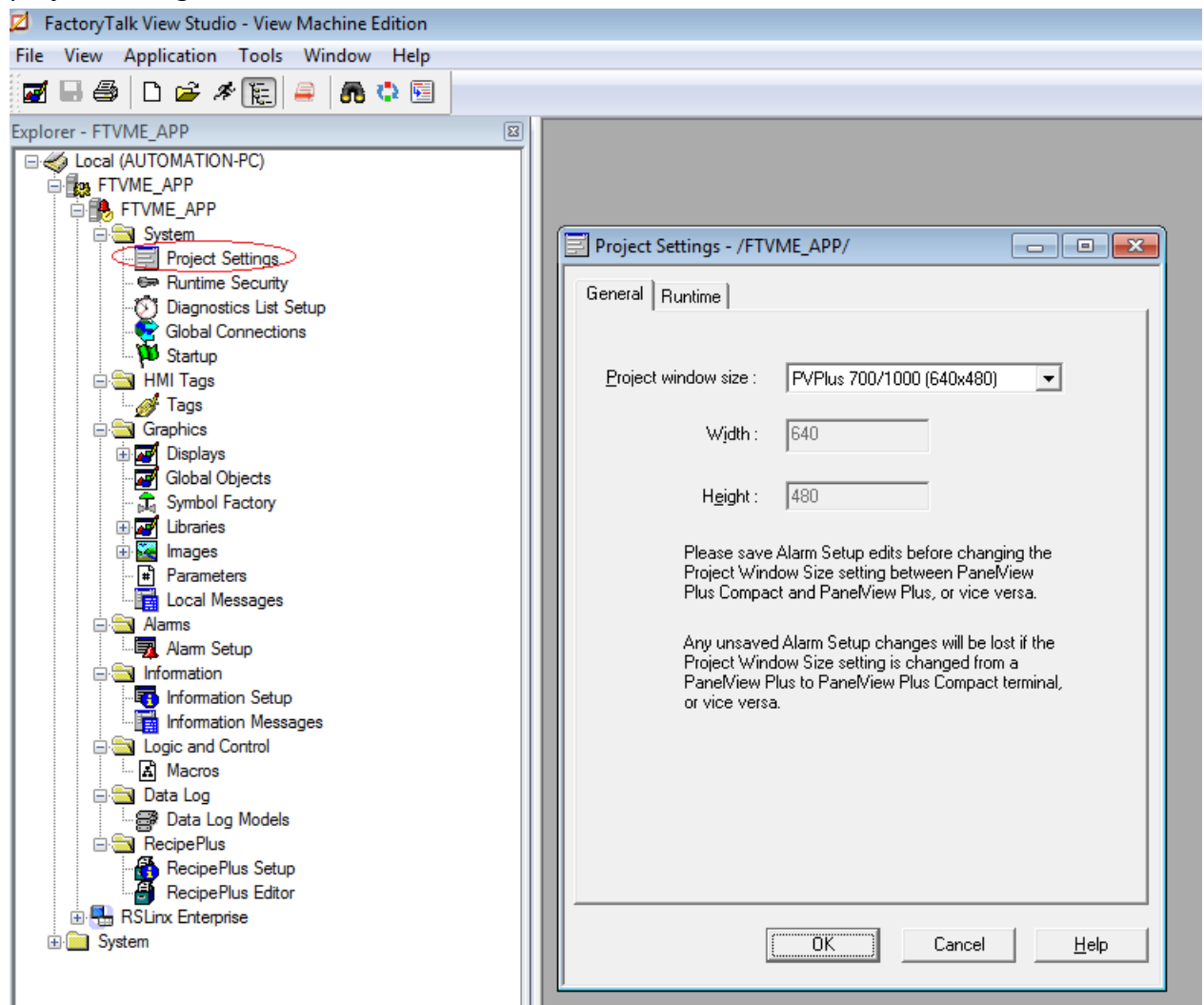


This now completes advanced lab 4. Please wait for instructions to proceed further.

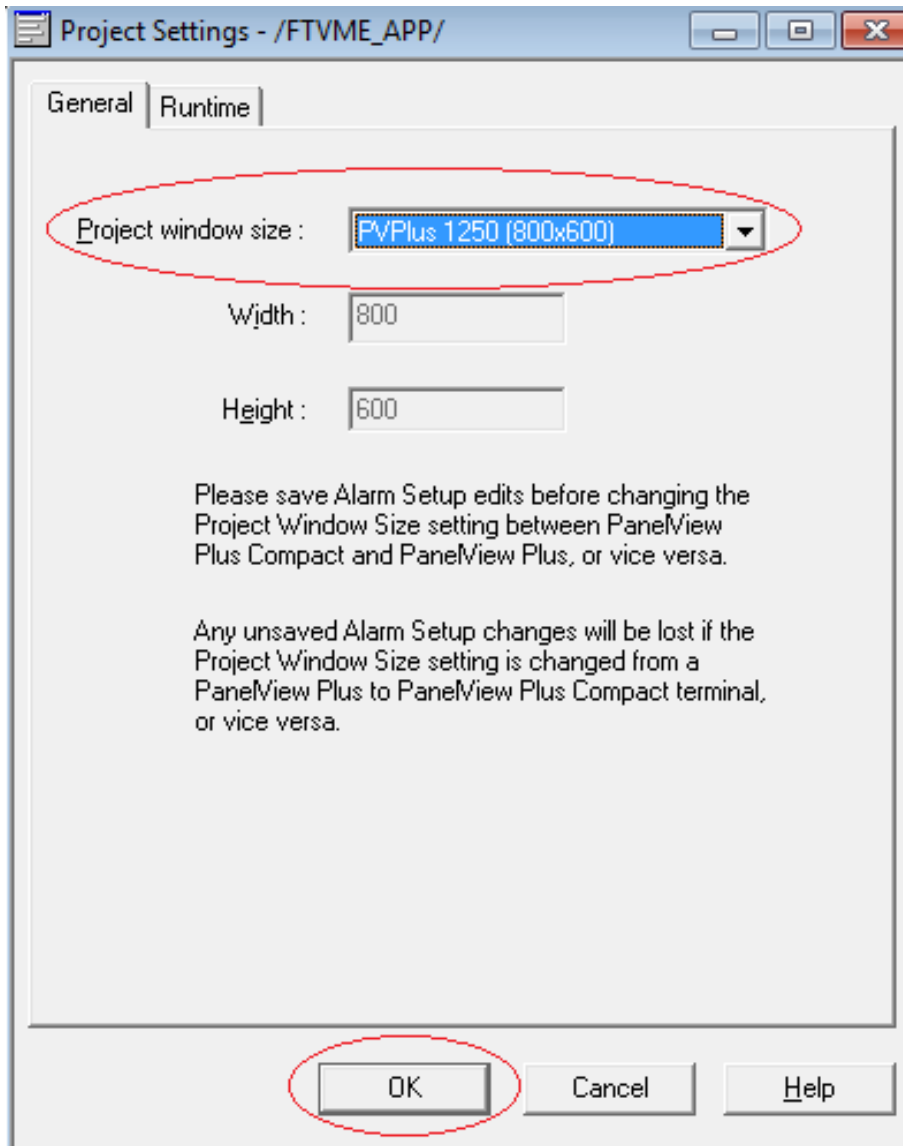
Advanced lab 5: Resizing the display

This lab is for demonstration purposes. Downloading the wrong size display could lead to unexpected results. Consult your user manual and part number of your PanelView before changing the project window size.

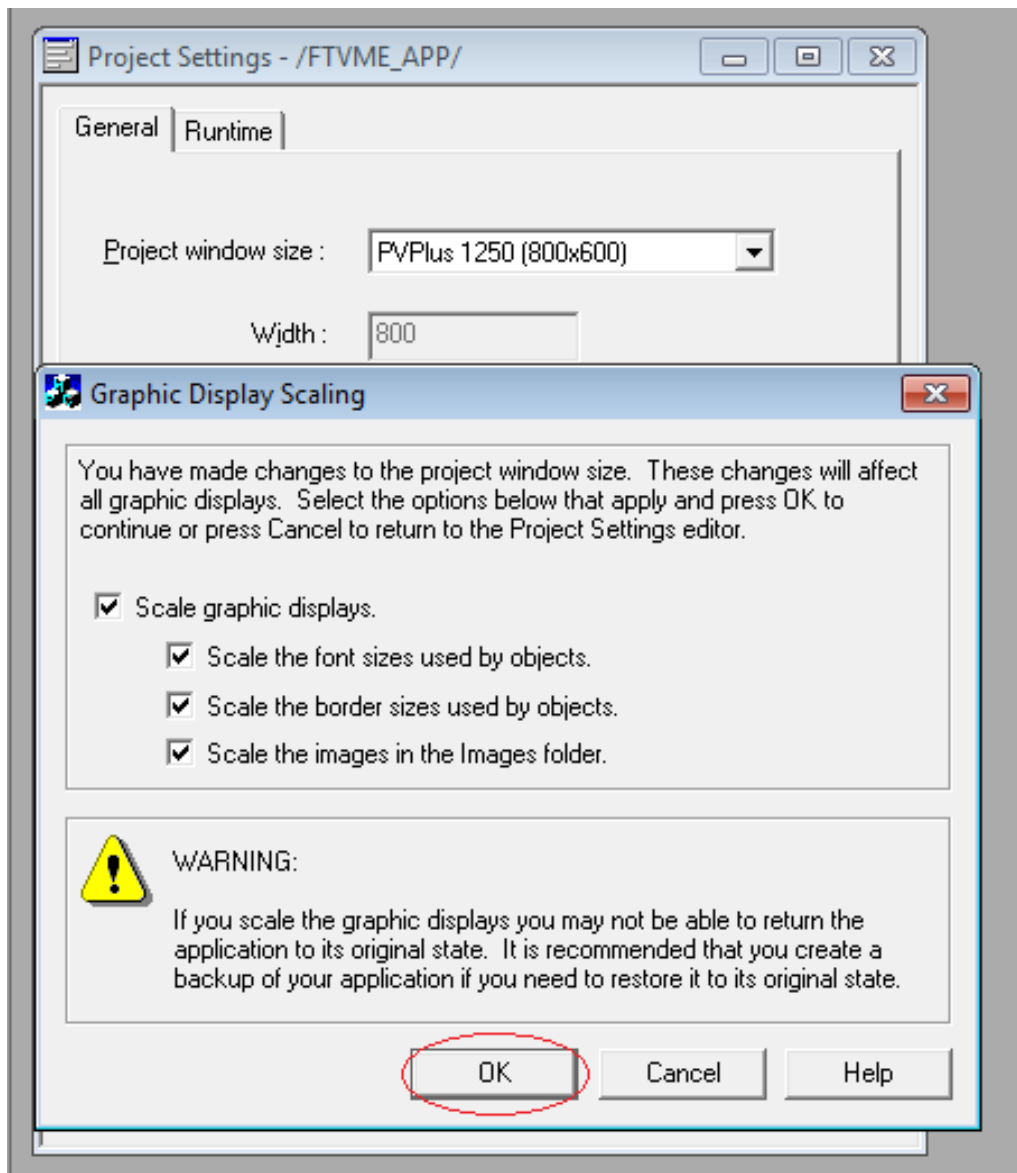
1. To resize the display project window size for a change of PanelView Plus screen size, start by opening the “FTVME_APP” in FactoryTalk View Studio if it is not already open. From the project explorer, double-click the “Project settings” link. This will open the project settings window.



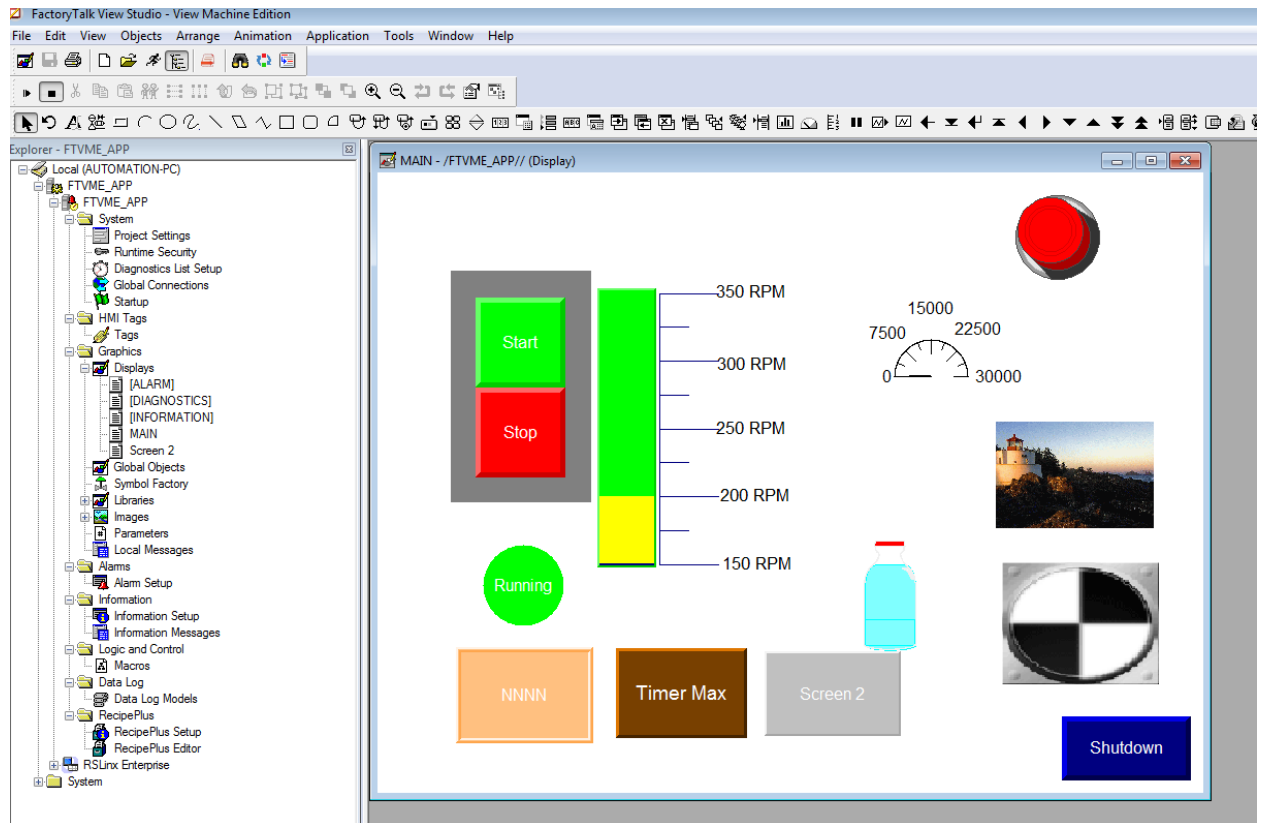
2. From the project settings window, click the drop down menu for “Project window size” and change it from PVPlus 700/1000 to PVPlus 1250. Once you have made this change, click the “OK” button at the bottom of the project settings window.



3. After clicking the “OK” button from step 2, the “Graphic Display Scaling” window will open. Leave all the default boxes checked and take note of the warning message. Click the “OK” button at the bottom of the window.



- After clicking “OK” from step 3, a brief loading bar will appear and then all the windows will close. Open up one of your displays and note the size change of the display itself and all the objects.



- After changing the screen resolution in step 3, change to screen resolution back to “PVPlus 700/1000 (640x480)” so that we can move on to the next lab.

Advance Lab 5 review:

1. What other changes can you make to your program from the project settings menu?
2. When making changes to your display resolution, would you ever have to worry about the scaling of graphics?

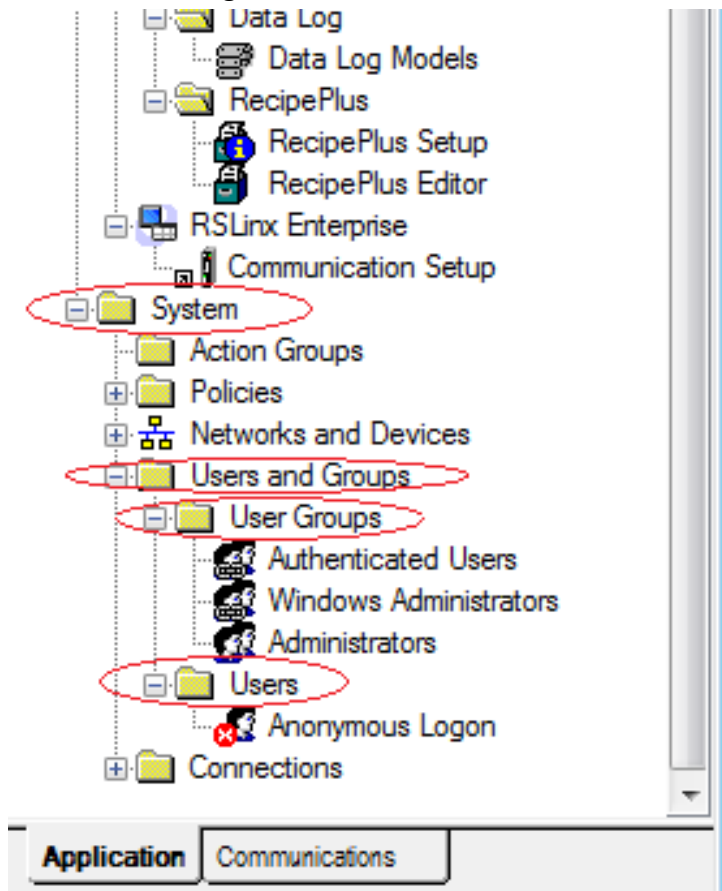


This now completes advanced lab 5. Please wait for instructions to proceed further.

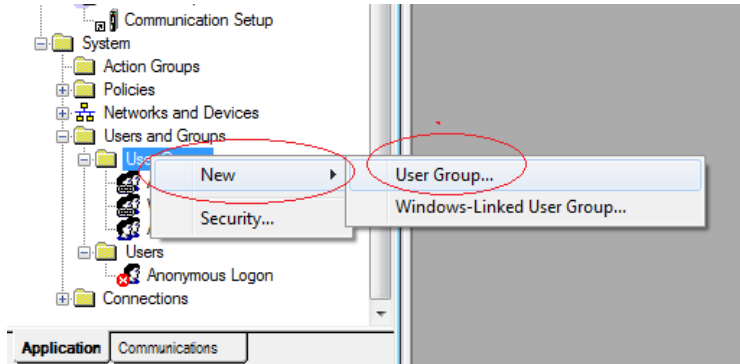
Advanced Lab 6 part A: Configuration of FactoryTalk Users and Groups.

Objective: To help the user understand how users and groups are configured and used within a Factory Talk View Studio machine edition application. By the end of this lab, the user should have a good understanding of how to add users and groups and also be able to configure credentials so that they can establish a basic level of security within their application.

1. To begin the configuration of FactoryTalk users and groups, we will start by locating the section of the project explorer that contains the default users and groups. Start by going to the project explorer and expanding the system folder, then the Users and Groups folder, then the User groups folder, followed by the Users folder. Your progress should look like the image below.

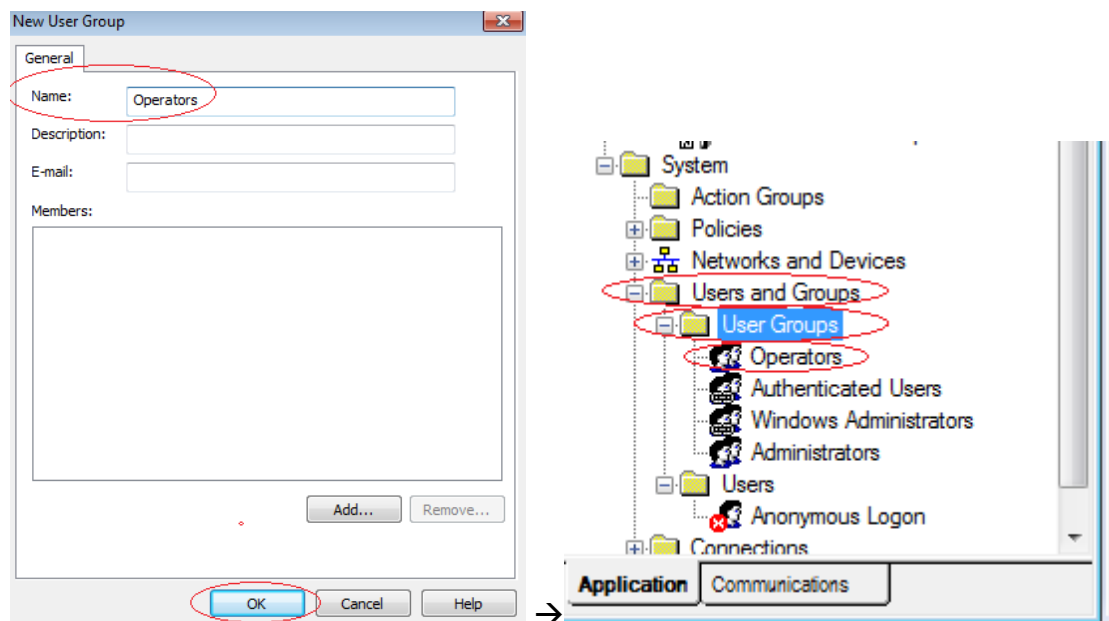


- Now that we have the “User Groups” & “Users” folders expanded, we can begin the process of adding users and groups to our application. In this step we will be adding a couple different groups. Start by right-clicking on the “user groups” folder and selecting “New→User Group...”

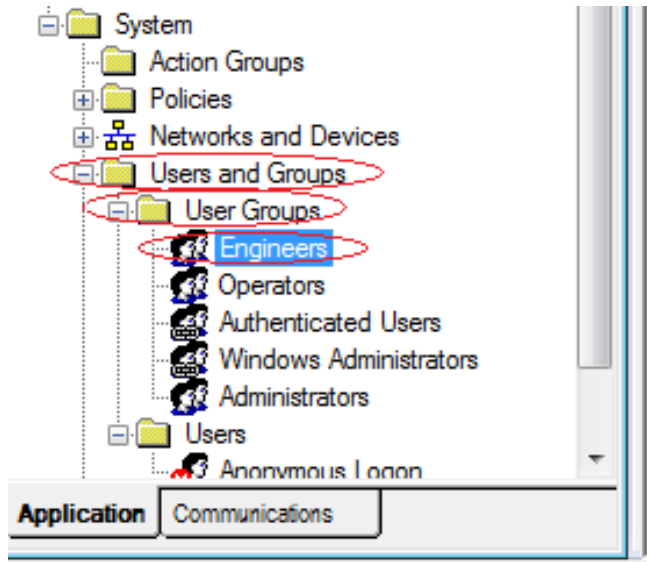


- After choosing to create a new user group from step 2, a window will open, from which you will configure a new group. There will be some information we add into the fields of this window. Enter the following information into the “New User Group” window:
 - Name: Operators

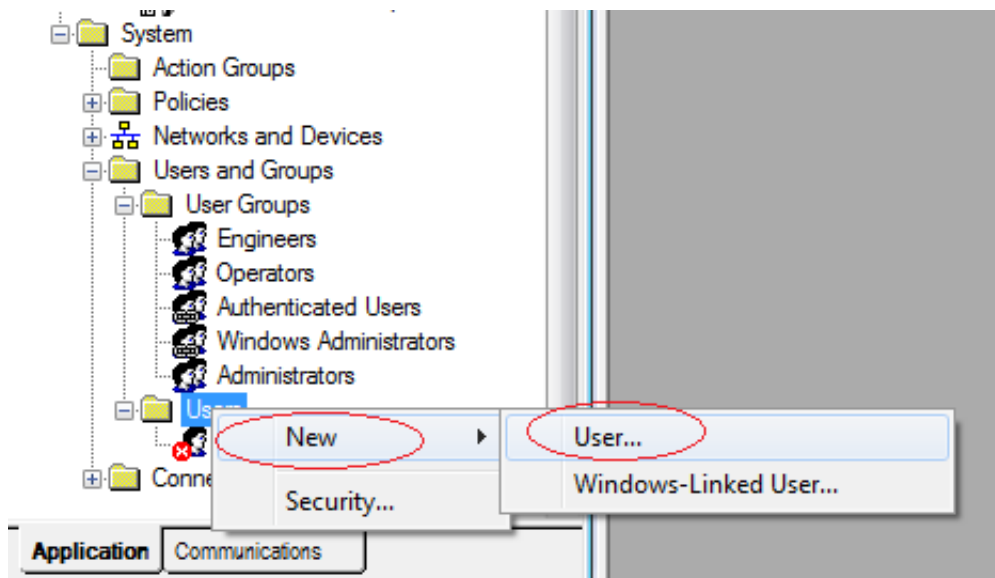
Once you have entered in the information listed above, click the “OK” button at the bottom of the window. If you made these changes successfully, you will see your new User Group underneath the user groups section of the project explorer as shown below:



4. After successfully creating the new user group that we named “Operators”, repeat steps 2 & 3 to create the additional user group “Engineers”. Once you have successfully added this group, your user groups section of the project explorer will look like the image below.



5. Now that we have added a couple new user groups, we can start adding users that will be members to those groups. To add a new user start by right clicking on the users folder within the project explorer, then select “New→User...”. This will open up the new user window.



6. From the new user window, the default tab that will be open is the “General” tab. From the general tab, there are a couple changes we need to make. Enter the following information into the general tab of the new user window.

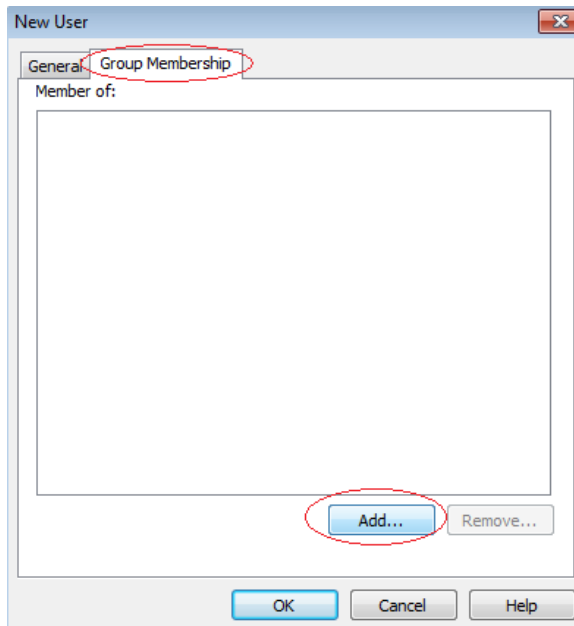
- User Name: Bob
- Check the box for “Password never expires”
- Enter the password of “rockwell”, all lower-case
- Enter the same password in the confirm field

Once you have successfully made these changes, your general tab of the new user window should look like the image below. After making the changes, click over to the “Group Membership” tab.

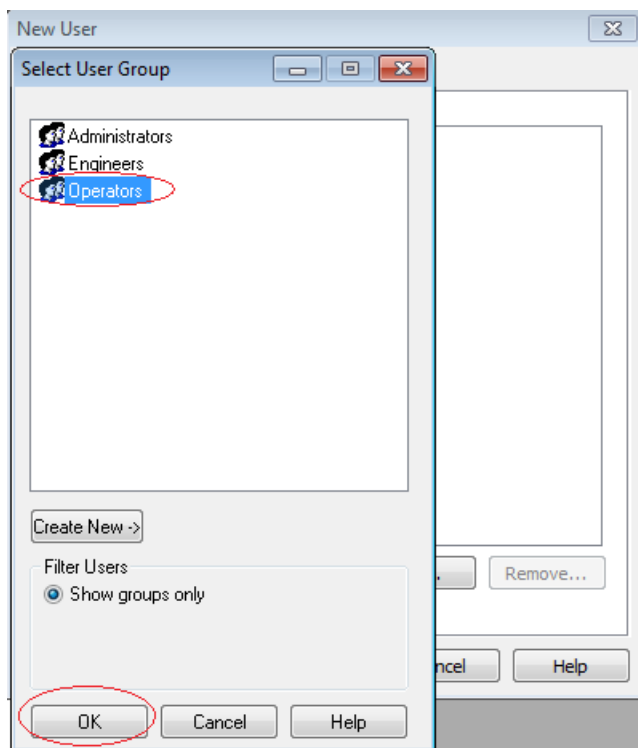
The screenshot shows a 'New User' dialog box with two tabs: 'General' and 'Group Membership'. The 'General' tab is active. The 'User name' field is filled with 'Bob'. The 'Full name', 'Description', and 'E-mail' fields are empty. The 'Password never expires' checkbox is checked. The 'Password' and 'Confirm' fields are filled with masked characters (dots). The 'OK', 'Cancel', and 'Help' buttons are at the bottom.

Field	Value
User name:	Bob
Full name:	
Description:	
E-mail:	
User must change password at next logon	☐
User cannot change password	☐
Password never expires	☒
Account is disabled	☐
Password:	
Confirm:	

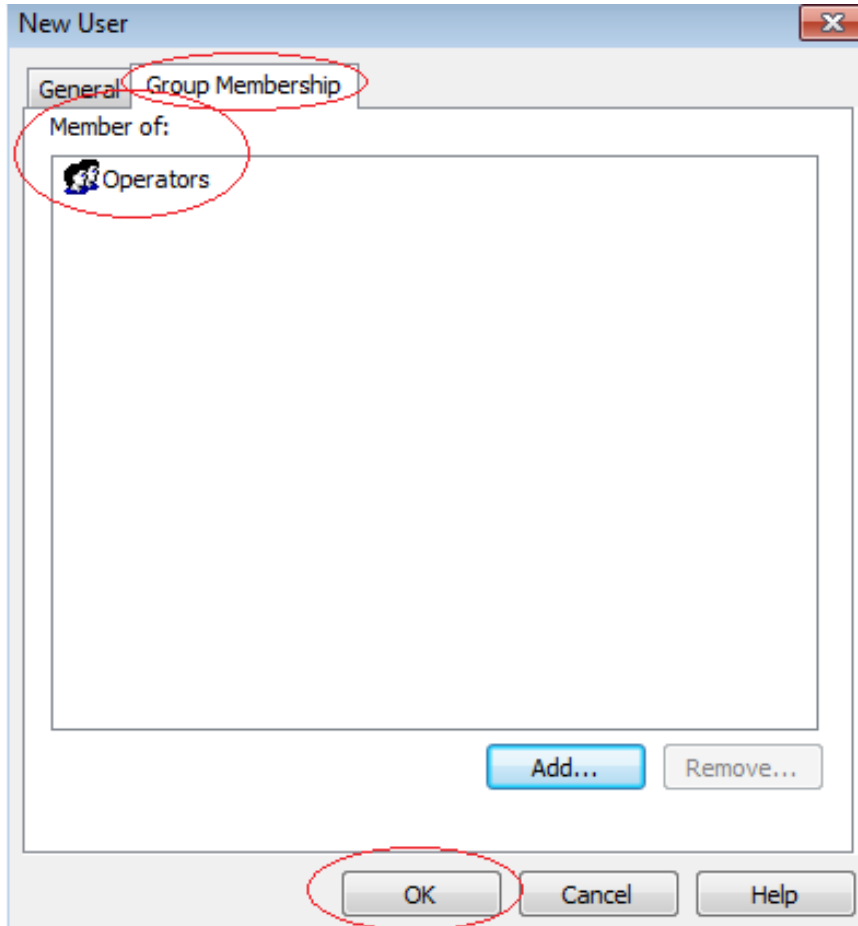
7. From within the group membership tab, there will be no groups listed by default. Click the “Add...” button at the bottom of the window to browse a list of groups the our user can be a member of.



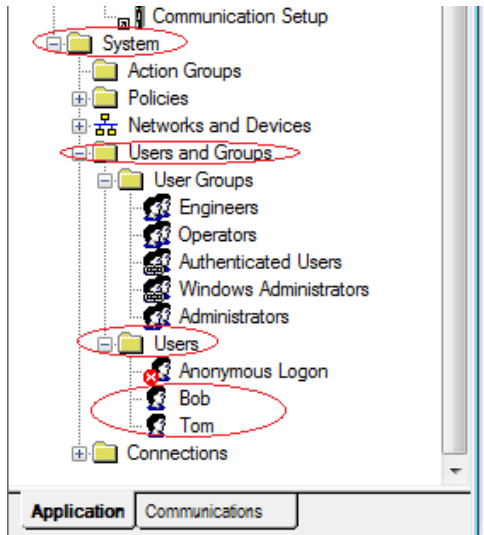
After clicking the “Add...” button from the image above, you will get a “Select user group” window as shown in the image below. Select the “Operators” group, then click the “OK” button at the bottom of the window.



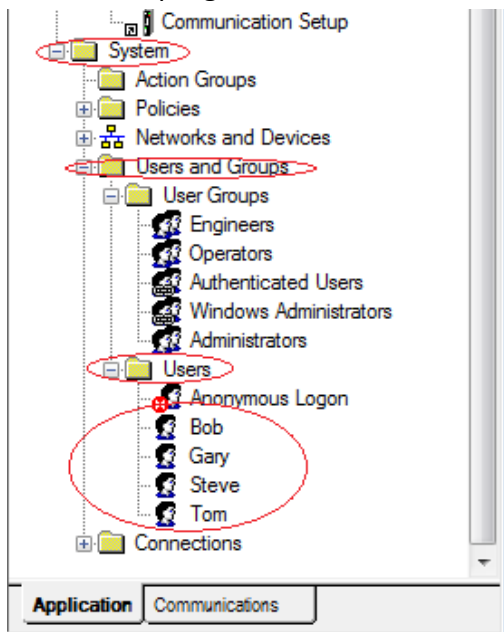
After you click the “OK” button from the image above, you will be brought back to the new user window, this time showing that our user “Bob” is now a member of the “Operators” group. Click the “OK” button at the bottom of the new user window and the new user window will close.



8. Now that we have successfully added “Bob” to the user group “Operators”, create another identical user named “Tom” with the same password and also a member of the operators group. Once completed, your “Users” section of the project explorer should look like the image below.



9. Through step 8 of this lab, we have created a couple groups and a couple users. The “Engineers” group still does not have any users assigned to it. Create two new users, one named “Steve” and the other “Gary”. Make both of these users members of the engineers group. Use steps 5 through 7 to guide you through this process if you get stuck. Your progress should look like the image below.



Advanced lab 6 review:

1. Where would I configure settings so that a user's password has to be changed upon first log in?
2. Can Windows user accounts be linked in to our FactoryTalk application?
3. Can a user be part of more than 1 group at a time?

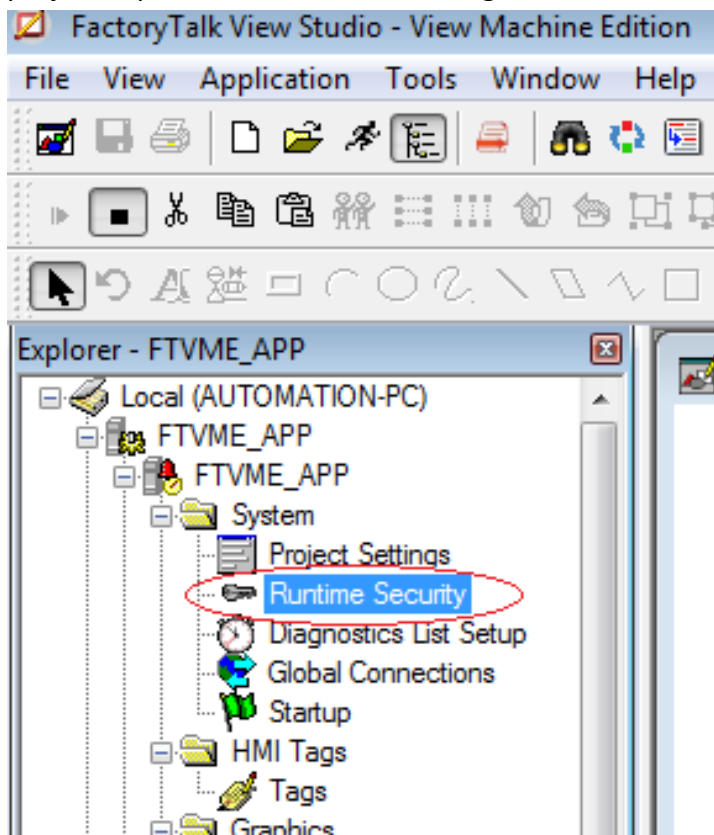


This now completes advanced lab 6 part A. Please wait for instructions to proceed further.

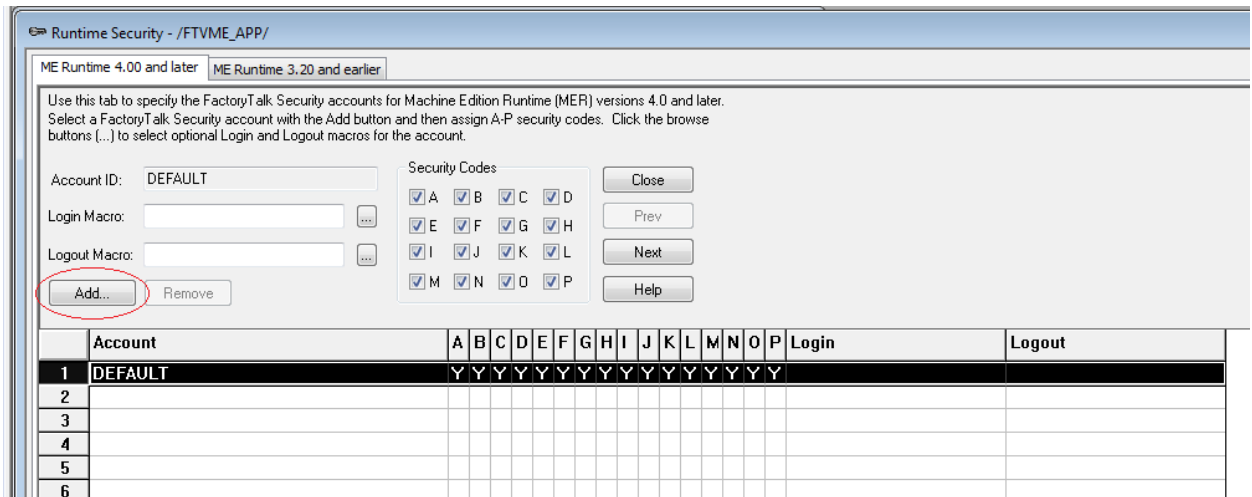
Advanced lab 6 part B: Screen security

Objective: To get the user familiar with configuring screen security that directly correlates with the FactoryTalk users and groups from part A of this lab. By the end of this lab, the user should have a good understanding of page security codes, assigning users and groups access to these codes, and be able to implement this process within their own day to day jobs as needed.

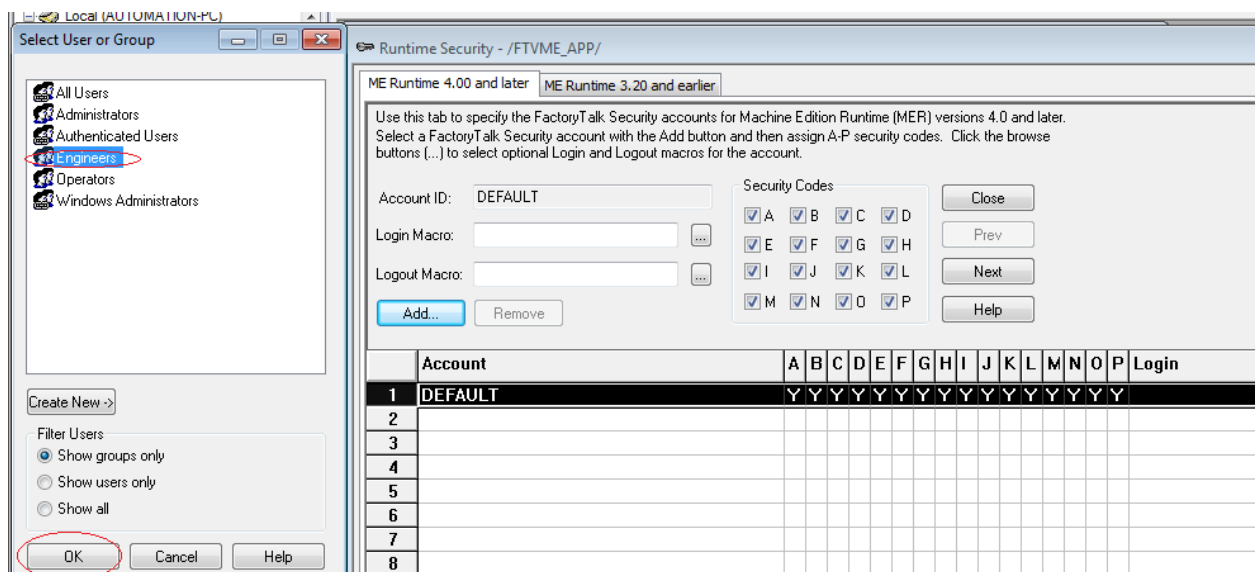
1. To set up screen security that will work along with the users and groups we created in the previous lab, we need to start by configuring the “Runtime Security” page. To access this configuration page, start by double-clicking on the “Runtime Security” link in the project explorer as shown in the image below:



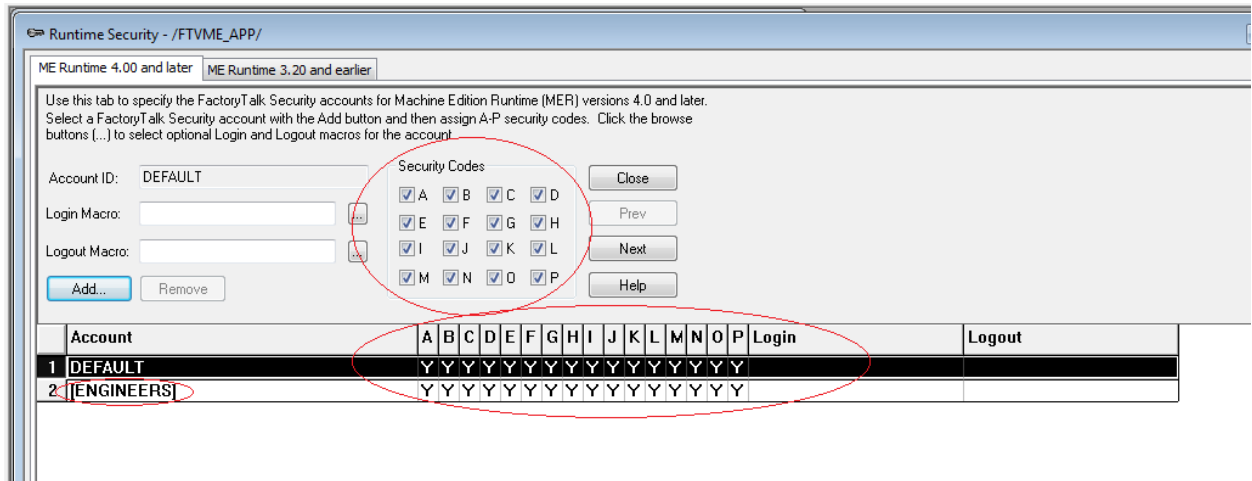
- After completing step one, the runtime security window should now be open for configuration. The first thing we will do within this window is add our user groups from the previous lab. Start by clicking the “Add” button.



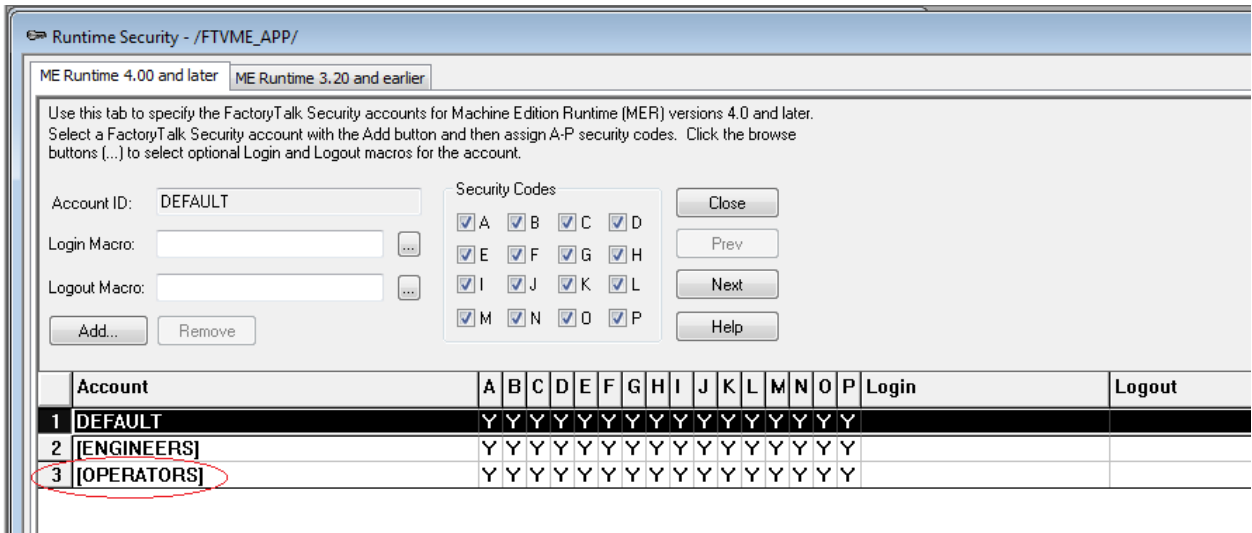
- After clicking the “Add...” button from step 2, you will be prompted to select a user or group to add to the account list. From the list, select the user group “Engineers”, then click the “OK” button at the bottom of that list.



4. After clicking the “OK” button from the previous step, your progress should now look like the image below. By default, all security codes (Letters A through P) will be enabled. We will leave all the codes enabled for the sake of this lab. You will notice also that the engineers group in the account list has a “Y” for each security code. This is because the engineers group will have access to screens with all of the security codes.

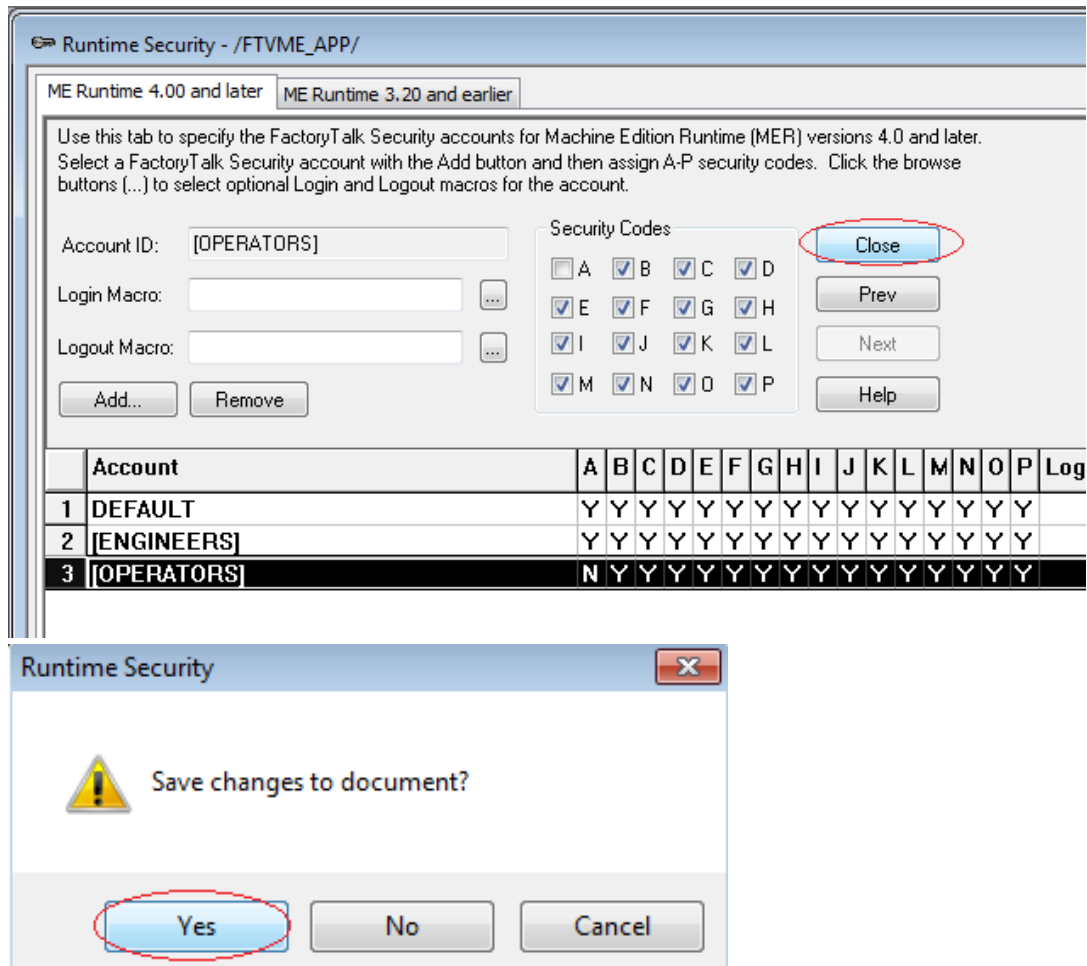


5. Repeat steps 2 and 3 to add the user group “Operators” to the account list. Once you have successfully made these changes, your progress should look like the image below.

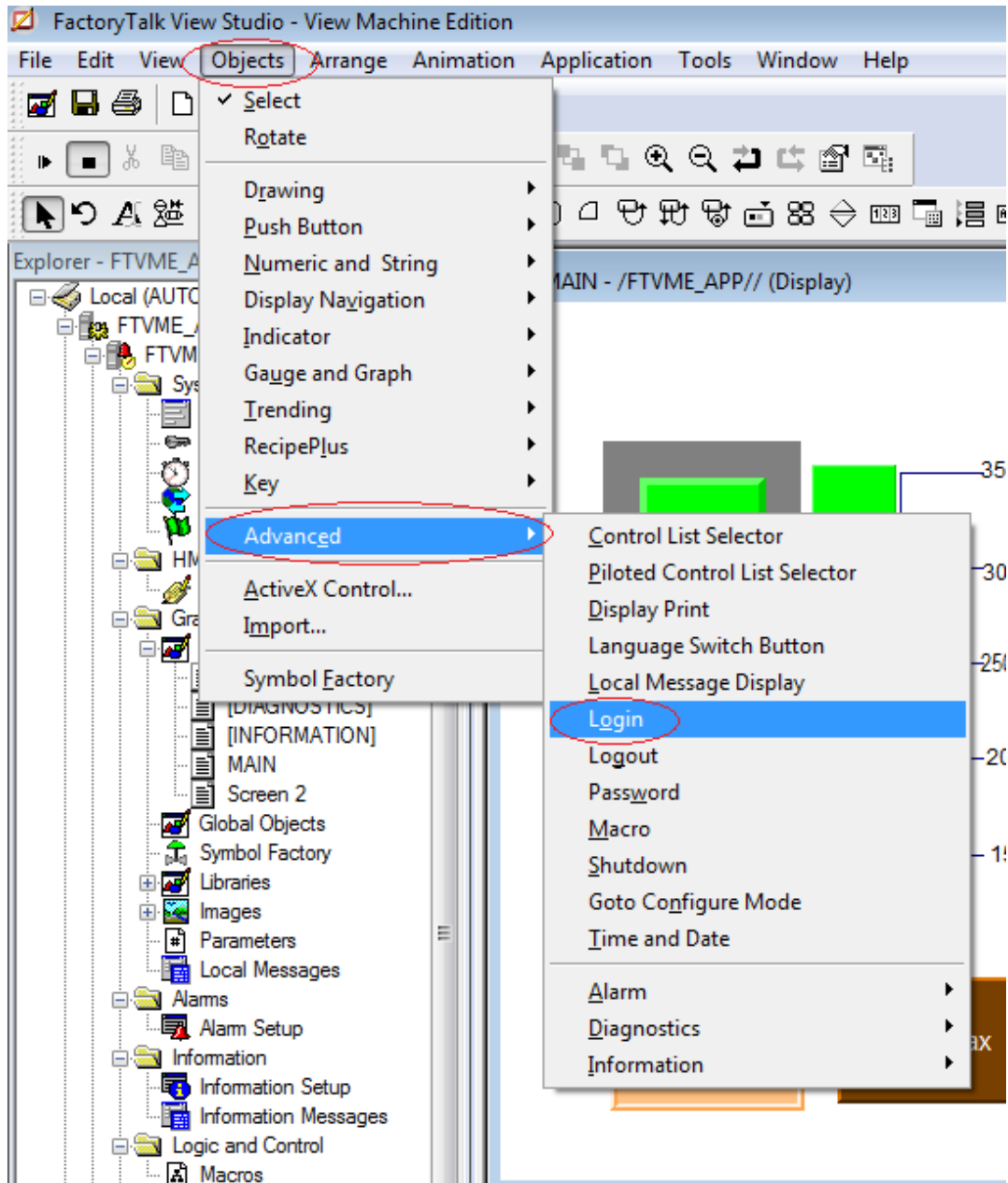


-
- The screenshot shows the "FactoryTalk Security Accounts" dialog box. The title bar reads "Runtime Security - /FTVME_APP/". There are two tabs at the top: "ME Runtime 4.00 and later" (selected) and "ME Runtime 3.20 and earlier".
- Below the tabs, there is instructional text: "Use this tab to specify the FactoryTalk Security accounts for Machine Edition Runtime (MER) versions 4.0 and later. Select a FactoryTalk Security account with the Add button and then assign A-P security codes. Click the browse buttons (...) to select optional Login and Logout macros for the account."
- The main area contains several fields and controls:
- Account ID:** A text field containing "[OPERATORS]".
 - Login Macro:** An empty text field followed by a browse button (...).
 - Logout Macro:** An empty text field followed by a browse button (...).
 - Add... Remove**: Two buttons below the macro fields.
 - Security Codes**: A group box containing a grid of checkboxes labeled A through P. Checkmarks are present for B, C, D, E, F, G, H, I, J, K, L, M, N, O, and P. A red circle highlights this entire section.
 - Close Prev Next Help**: Four buttons arranged vertically on the right side.
- At the bottom, there is a table listing configured accounts:
- | | Account | A | B | C | D | E | F | G | H | I | J | K | L | M | N |
|---|-------------|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | DEFAULT | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y |
| 2 | [ENGINEERS] | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y |
| 3 | [OPERATORS] | N | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y | Y |
- In this table, the first column (index), the Account name, and the 'A' column are circled in red. Additionally, the row for [OPERATORS] is highlighted with a thick black background.

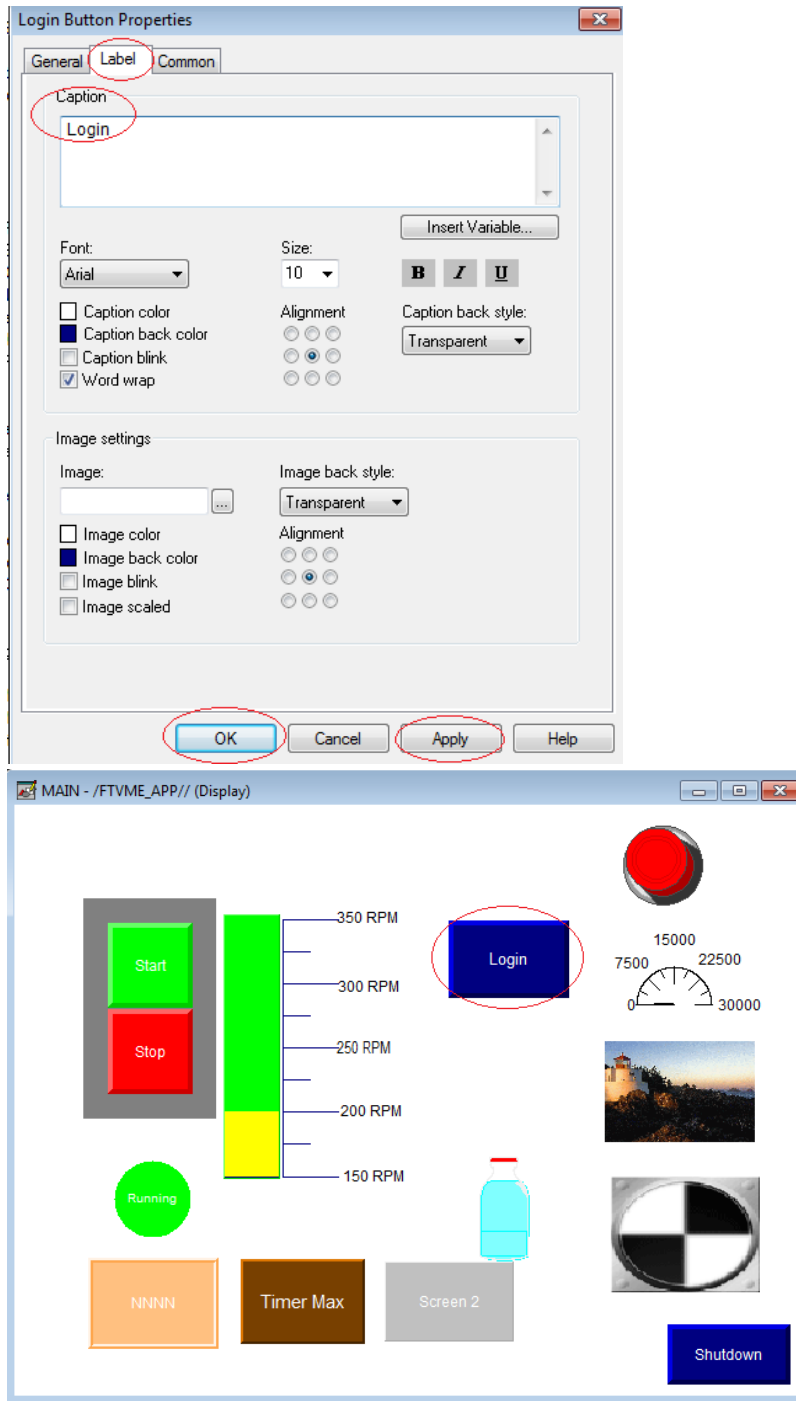
7. Now we need to save the settings we made to the “Runtime Security” page. This can be done by clicking the “Close” button as shown below. Also, when prompted to save changes to the page, click yes as shown below.



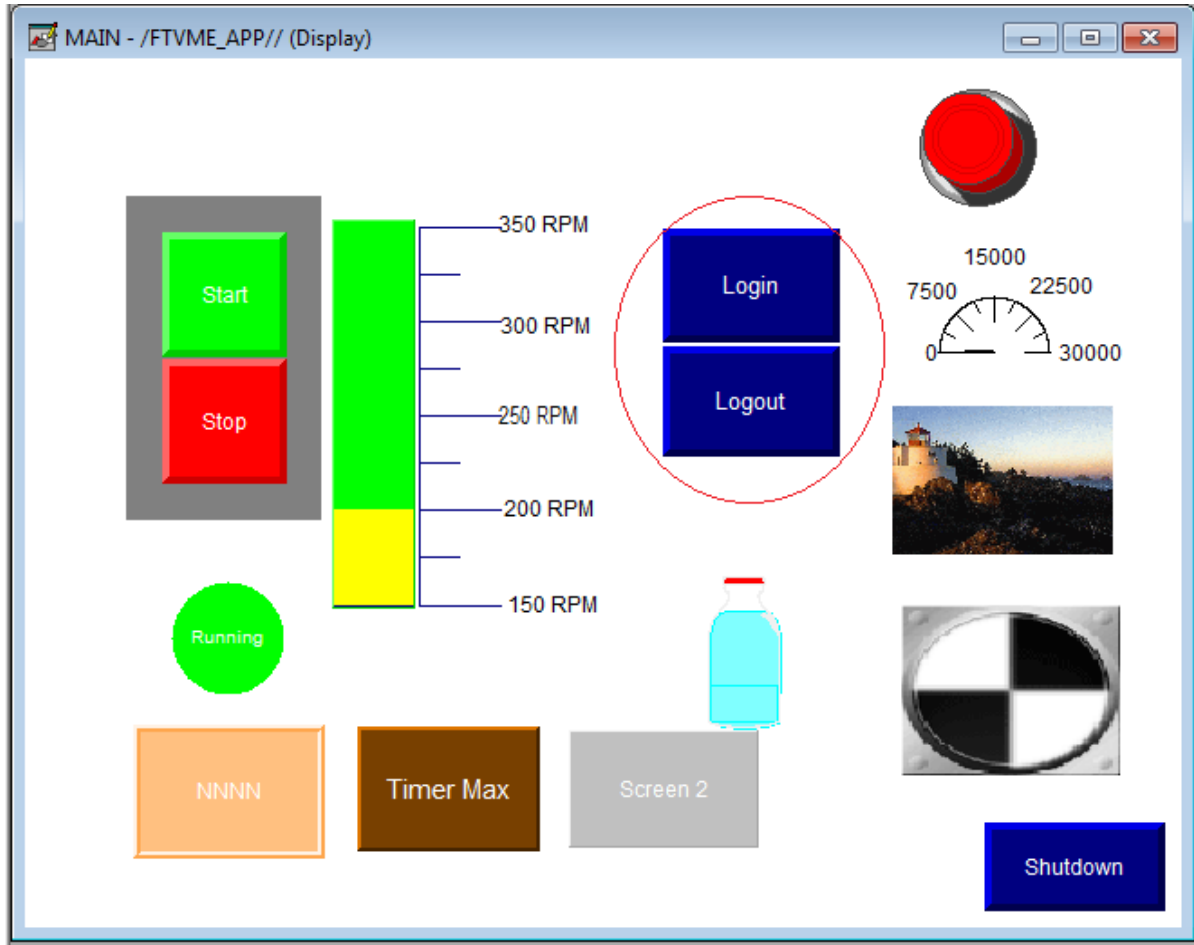
8. Now that we have made our changes to the user group security codes, we need to add a login and logout button on our main display so that we can test our new settings. Start by adding a “Login” button to an open space on your MAIN display. This button can be found by going to “Objects→Advanced→Login”.



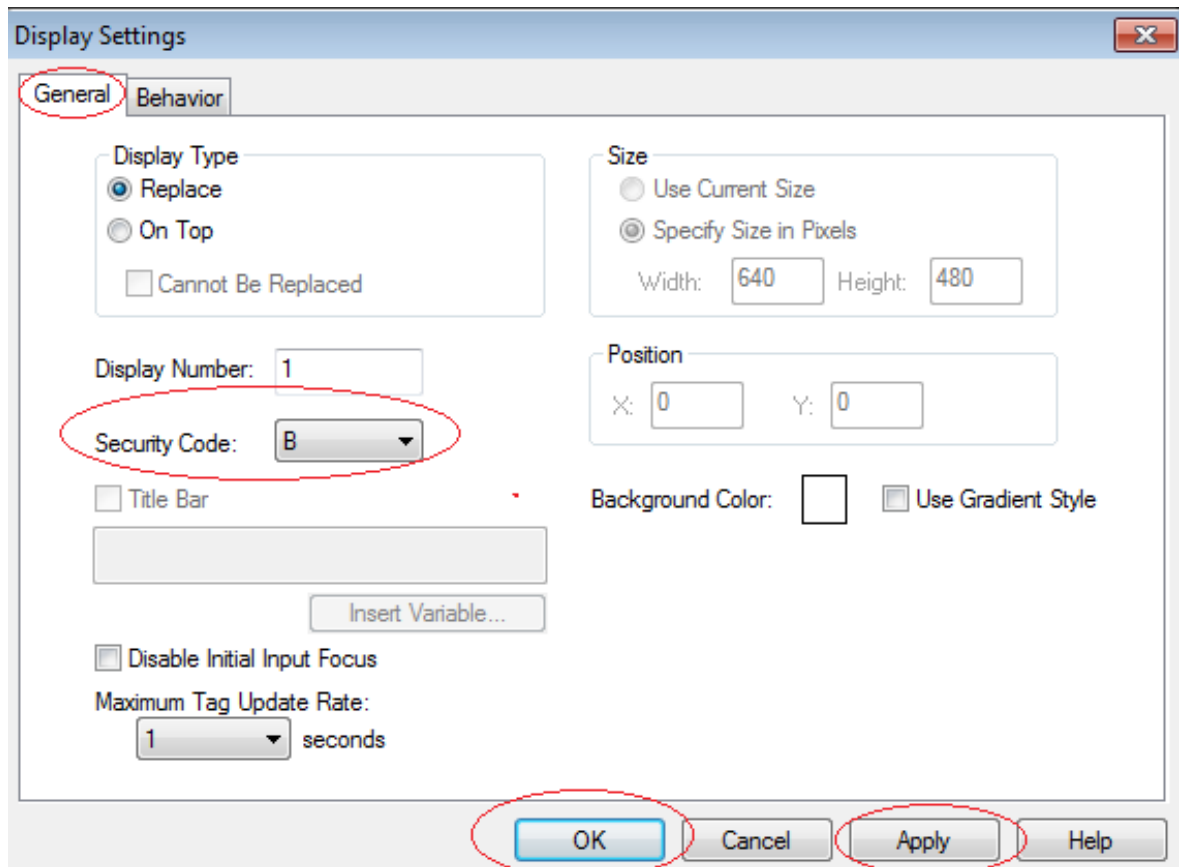
9. After selecting the “Login” button from step 8, draw the button onto the MAIN display in any open space. Once you draw the button, the properties window for the login button will open. Click over to the “Label” tab of the properties window and add a caption to the button of “Login”. After adding this caption, click the “Apply” followed by the “OK” button at the bottom of the properties window. Your progress should look like the images below.



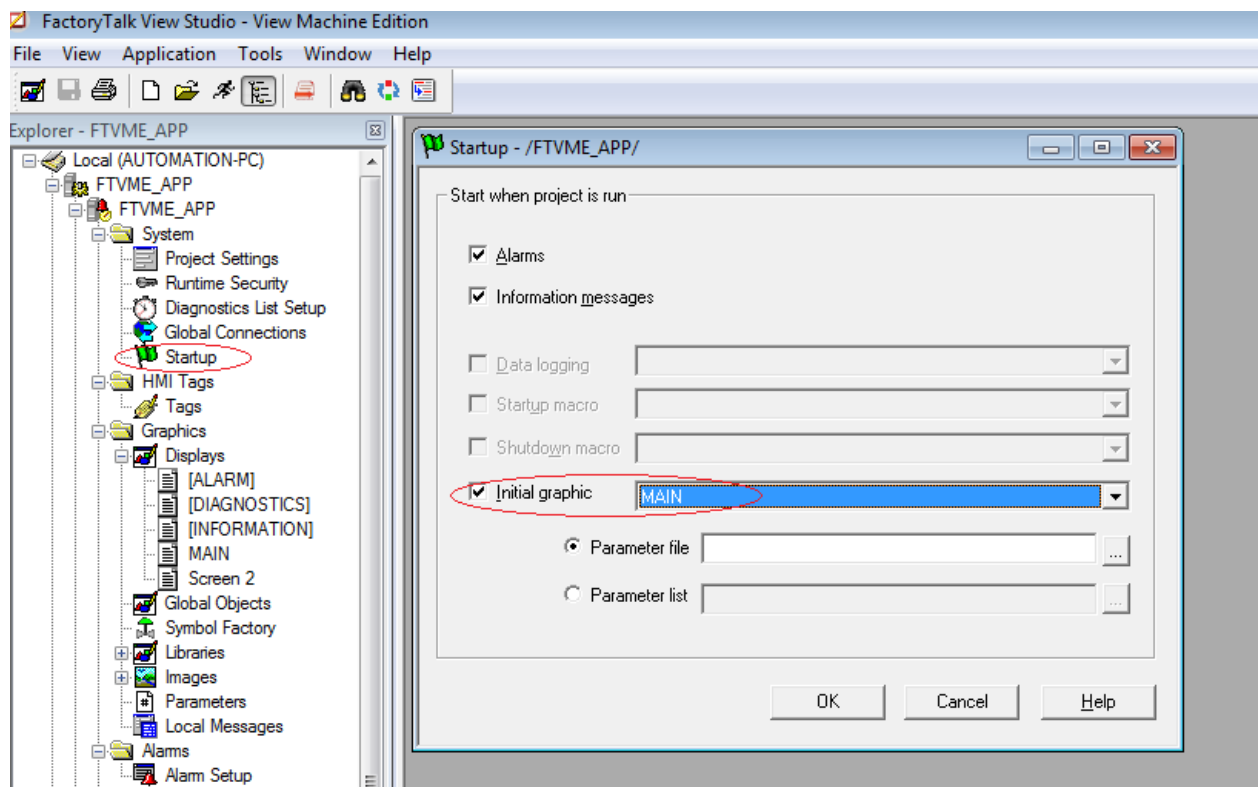
10. After completing step 9, you should have a “login” button on your main display. Repeat steps 8 & 9 but this time add a “logout” button directly below the login button on your MAIN display. After successfully making these changes, your MAIN display should look like the image below.



11. Now that we have added login and logout buttons through step 10, we need to assign security codes to our MAIN display and screen 2. Starting with the MAIN display, right-click on any open space within the display and select “Display Settings”. The default tab for display settings is where we will specify which security code this page will have. From the “Security code” drop-down menu, select security code “B”. Once you have this code selected, click the “Apply” follow by the “OK” button at the bottom of the display settings window. Your Display settings window for the MAIN display should look like the image below before proceeding to the next step.



12. Now we need to change security code settings for screen 2 so that only engineers have access to this screen. Repeat step 11 to change the security code settings of the “Screen 2” display, but this time assign security code “A” to screen 2.
13. The last step is to make sure that the MAIN display opens upon power-up since that is where our login and logout buttons are located. Start by double-clicking on the “Startup” link within the project explorer tree. Within the “Startup” window, specify the “MAIN” display as the initial graphic. Once you have made this change, click the “OK” button at the bottom of the startup window.



14. Save all changes to the MAIN and screen 2 displays. Create a new .MER file and download to your PanelView. Make sure you can login and logout as various engineers and operators with the login and logout buttons. Also, make sure that if you are logged in as an engineer, you can access screen 2. If you log in as an operator user, make sure your access to screen 2 is denied.

Advanced lab 6 part B review:

1. What would be another way to log different users in, other than a login / logout button?
2. Can I specify security code security at an individual user level?
3. If I wanted a macro to run for a given login, where would that be configured?

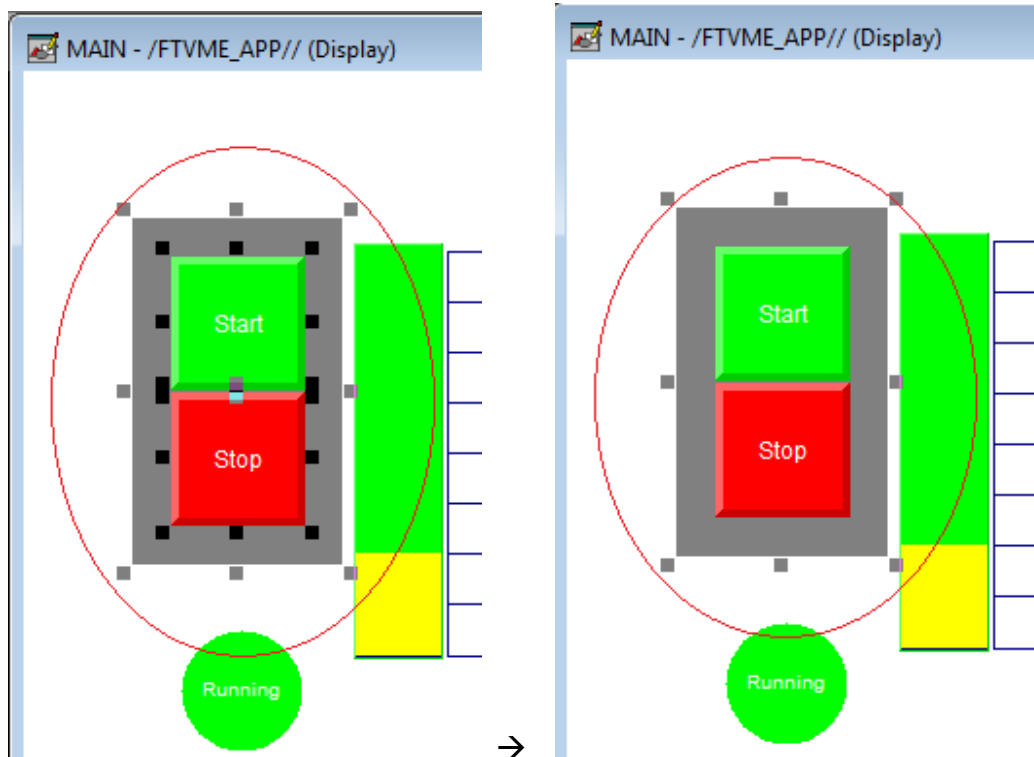


This now completes advanced lab 6 part B. please wait for instructions to proceed further.

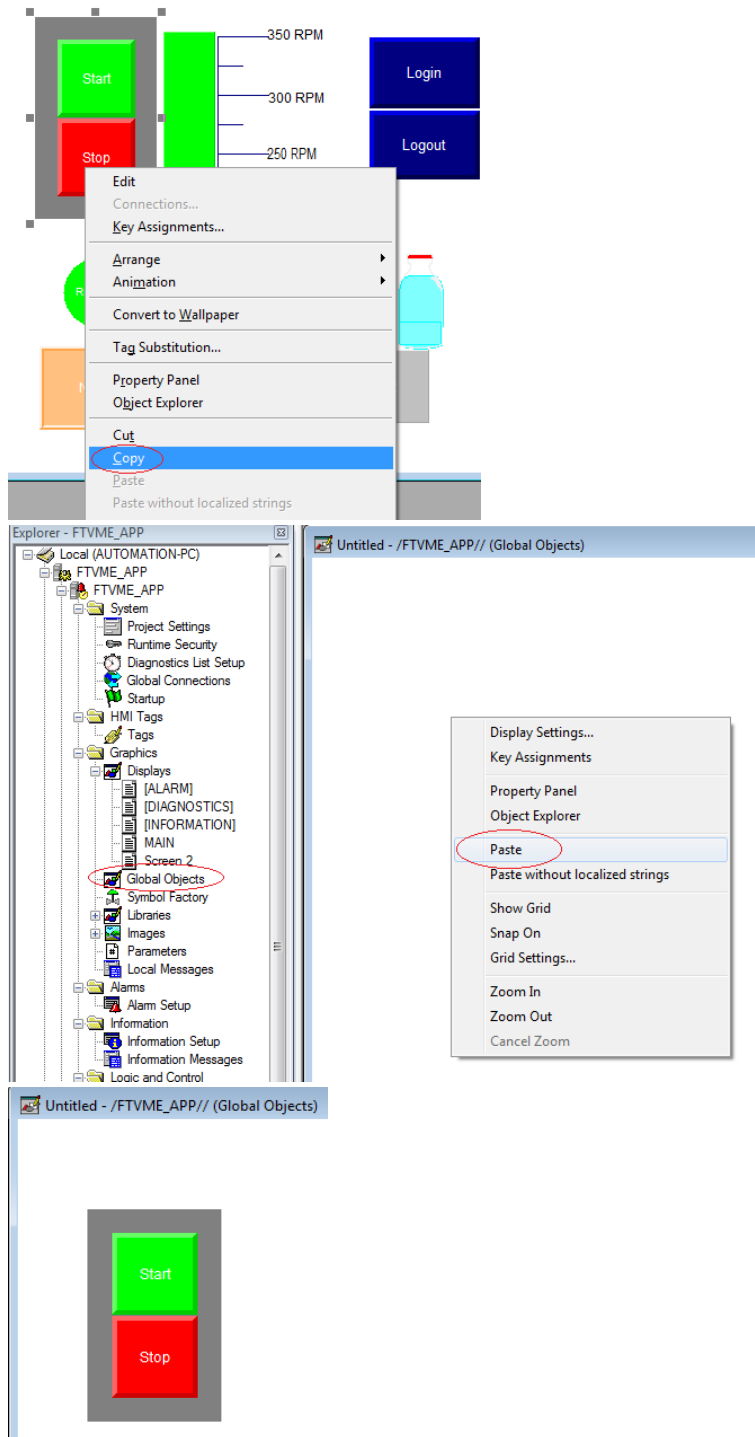
Advanced lab 7: Global Objects.

Objective: To familiarize the user with the use of global objects. By the end of this lab, the user will have a good understanding of what a global object is, how to configure one, and how to implement them in future projects if needed.

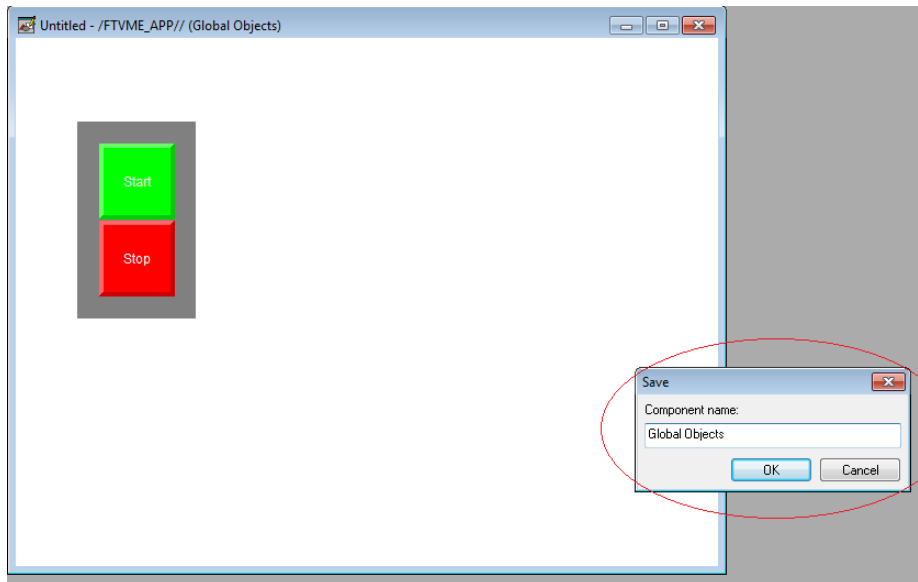
1. To start this lab, make sure your MAIN display is opened. From your MAIN display, highlight the start and stop button along with the rectangle behind the buttons. With all the objects highlighted, hit Ctrl+G on your keyboard, which will group these objects together as one. Your progress should look like the images below.



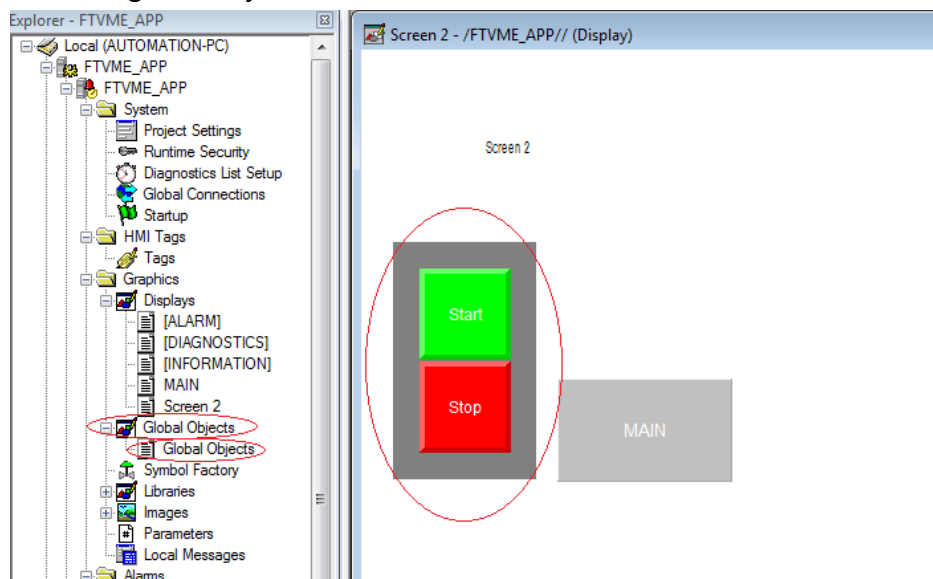
- Now that we have grouped the start/stop buttons together as one object, we will use this as an example of a global object. Right-click on the newly grouped object and select copy. Open the “Global Objects” window by double clicking on the link within the project explorer. Once the global objects window opens, paste this object into the global objects display. Your progress should look like the images below.



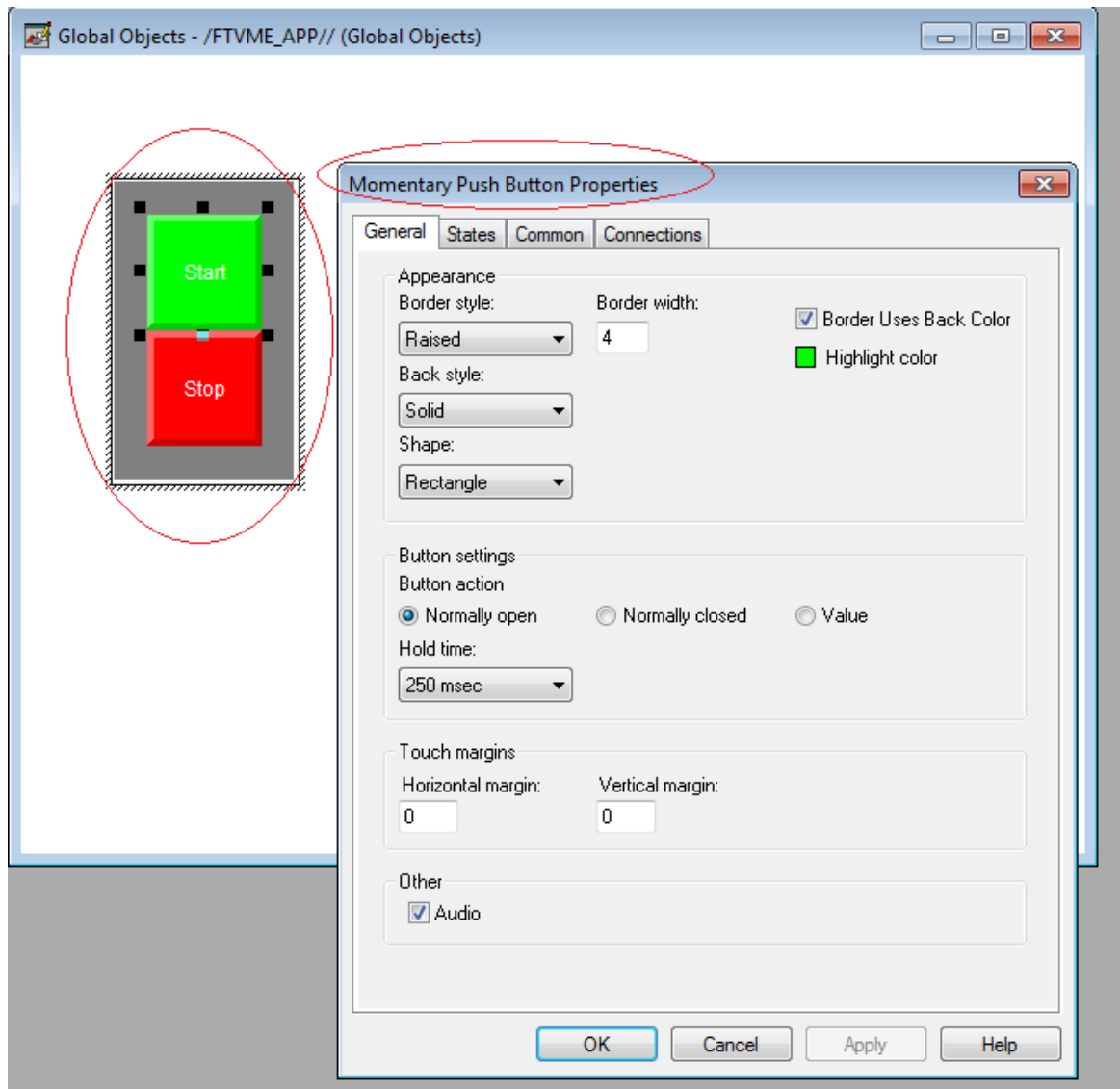
- Click the red X located at the top of the global objects display and save changes when prompted. You will be asked to rename this display since it doesn't have a name by default. Give the global objects display the name of "Global Objects".



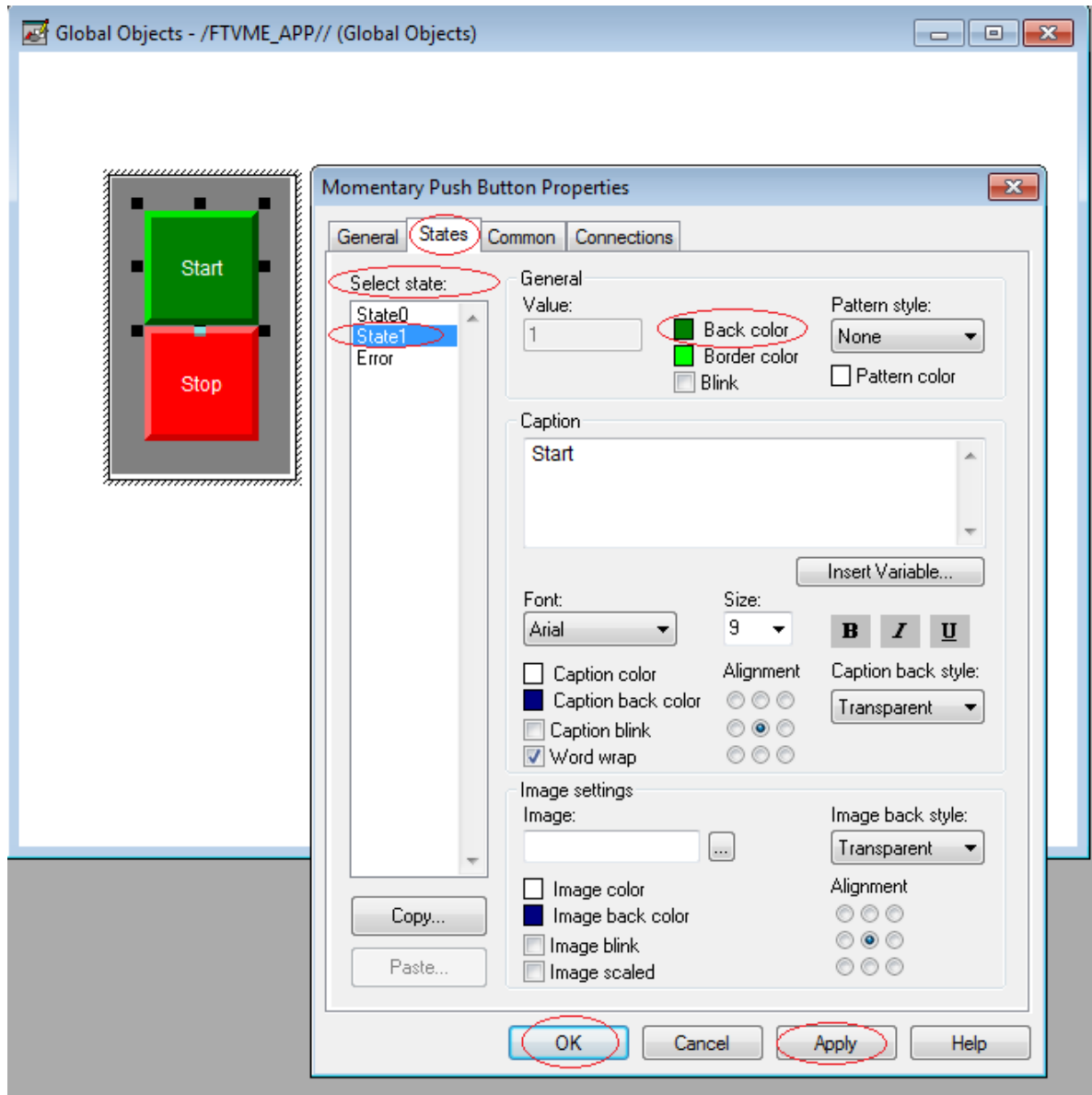
- Now that we have made the start/stop station a global object, we can start adding it to the screens that we know it will be on. Since we used an existing object to create a new global object, we will need to delete the original start/stop station on the MAIN display and replace it with the global object start/stop station. Open up the newly created "Global Objects" display, right-click on the start/stop station and paste it onto screen 2 as shown below. Also, make sure you replace the MAIN display start/stop station with this new global object.



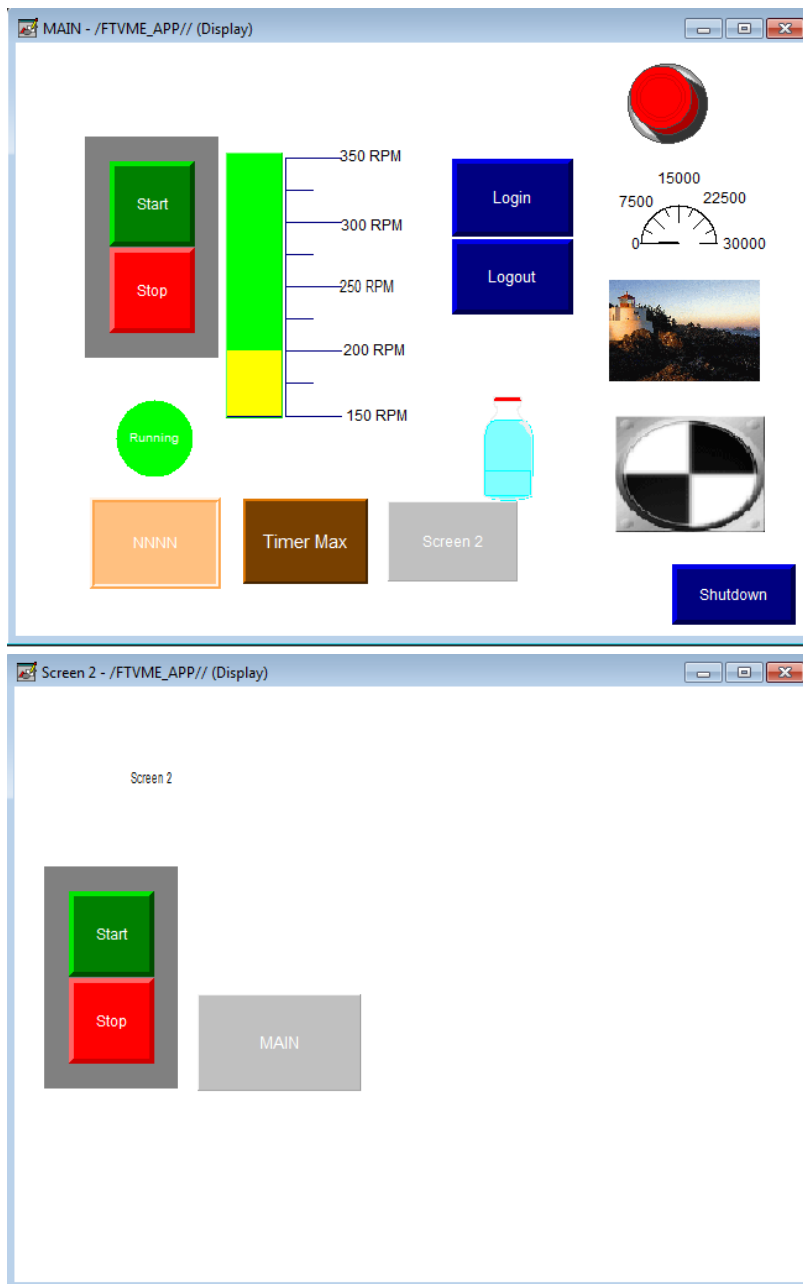
- After completing step 4 we should now have the start/stop station that we made a global object, on the MAIN and screen 2 displays. Start by saving changes to all displays, then closing all open displays except for the global objects display. From the global objects window, we will be making changes to the start/stop station that we are using on multiple displays. We will make changes from the global objects window so that those changes are pushed out to every instance of that object within the program. Click on the start button of the global object until you get into the properties of that individual button as shown below.



6. From within the start button properties window, select the “States” tab. From within the states tab, highlight “State 1” from the “Select state” list. Once state 1 is highlighted, change the back color of the button to dark green. Once you have made these changes, click the “Apply” followed by the “OK” button at the bottom of the properties window.



7. After clicking apply and ok from step 6, you will notice that the color of your start button will have changed to a dark green. Save the changes to the global objects display and open your MAIN display. You should now see that the original light green color has changed to a dark green. Open the “Screen 2” display and the same thing will have changed. This is extremely time saving when using the same objects on multiple screens. Play around with this same process and make other changes to that same global object. Also, try making other global objects using the same steps from this lab.



Advanced lab 7 review:

1. Why is using something like global objects useful?
2. What are the hot-keys for grouping and un-grouping objects?
3. What is the process for making a global object?



This now completes advanced lab 7. Please wait for instructions to proceed further.

Appendix A

Helpful Links:

Werner Electric's Website: All things Werner electric. Find contact information, product lines, place orders and much more! <http://www.wernermn.com/>

Rockwell Automation Knowledgebase: Find technical notes from cases that have been solved by Rockwell support engineers.

<https://rockwellautomation.custhelp.com/>

Rockwell automation's support page: Find software downloads, firmware updates, AOP's, anything to be downloaded from Rockwell Automation

[http://www.rockwellautomation.com/rockwellautomation/support/overview.
page?](http://www.rockwellautomation.com/rockwellautomation/support/overview.page?)

Rockwell automation literature library: Find all published documents from Rockwell including user manuals, technical data, product profiles, etc.

[http://literature.rockwellautomation.com/idc/groups/public/documents/web
assets/browse_category.hcst](http://literature.rockwellautomation.com/idc/groups/public/documents/web_assets/browse_category.hcst)

Rockwell automation product drawings / schematics: find CAD drawings and dimensions for all Rockwell products.

[http://www.rockwellautomation.com/rockwellautomation/support/selection.
page](http://www.rockwellautomation.com/rockwellautomation/support/selection.page)

Notes: